Vectorworks 2015 User's Guide

Table of Contents

Welcome

Welcome to the Vectorworks Help System

Welcome to Vectorworks® software—a CAD and BIM program that includes all the 2D and 3D tools and technology you need in one easy-to-use, cost-effective package. In addition to precision drafting, it provides powerful tools for 3D modeling that can be used to create, present, and even manufacture designs. Built-in database and worksheet capabilities help track costs and materials. A host of import and export formats makes it easy to share files.

The Vectorworks Fundamentals product includes extensive symbol libraries, default drawing resources, and a built-in scripting environment that can be used to automate routine tasks. Renderworks® is a plug-in product that allows you to create beautiful photo-realistic, hand-drawn, or schematic rendered views of your designs.

The Vectorworks Design Series software is a complete design and production management system for architects, landscape architects, lighting designers, and mechanical engineers. Vectorworks Design Series software includes the following products: Architect, Landmark, Spotlight, and Designer. Some features in the Vectorworks Design Series software expand upon core Vectorworks Fundamentals or Renderworks product functionality.

The Vectorworks Designer product includes the combined feature set of the Vectorworks Architect, Landmark, and Spotlight products. Therefore, it is not necessary to document each individual feature as belonging to the Vectorworks Designer product. The Designer workspace contains a major subset of the entire combined feature set.

Summaries of the new software features for all products are located in “New Features” on page 1. Descriptions of documentation improvements can be found in “Documentation Improvements” on page 18.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using the Help System

This help system serves as a comprehensive reference for the tools, commands, and dialog boxes contained in the Vectorworks Fundamentals and Design Series workspaces. The program is linked to the appropriate help topic, providing instant information.

Online Help and Locally-installed Help

To open the help system, select **Help > Vectorworks Help**. The help window opens in your default browser. If you are connected to the Internet, the online version of the help displays. If an Internet connection is not available, the locally-installed version of the help opens. The local version may not contain the latest information, but in general, the online and local versions are very similar.

If bandwidth or connection issues cause topic display or searching to become unacceptably slow, select the Vectorworks Session preference, **Use local help even when an Internet connection is available**, to always use the local version of the help system.

Using the local version of the help with the Internet Explorer browser may cause a blocked content warning. Enabling the blocked content is perfectly safe; however, the help opens to the first page in this situation. Search the help for the
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

item of interest at that time, switch to a different default browser, or use the online version of the help system to avoid this issue.

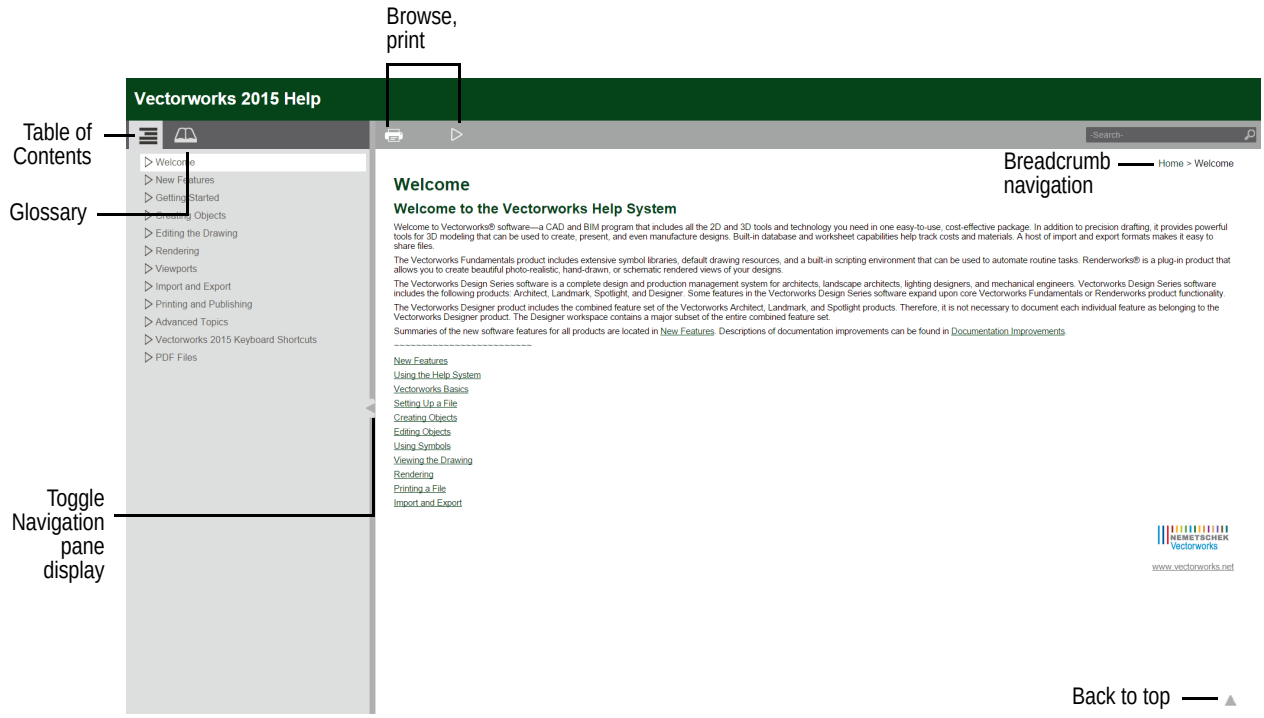
The Internet Explorer browser, prior to version 9, does not support the HTML5 features of this help system. Upgrade to a more recent version of the browser if possible, or switch to a different browser.

Help System Appearance

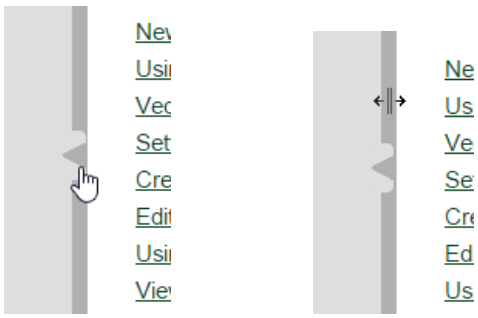
Depending on the size of your screen, the help system adjusts itself automatically. To display the help system on your mobile device, access the help from the Vectorworks | Support area of the website.

Desktop View

On a large screen, typically a desktop, the help opens from the **Vectorworks Help** menu command with an area for the Table of Contents and Glossary on the left, and the topics on the right.



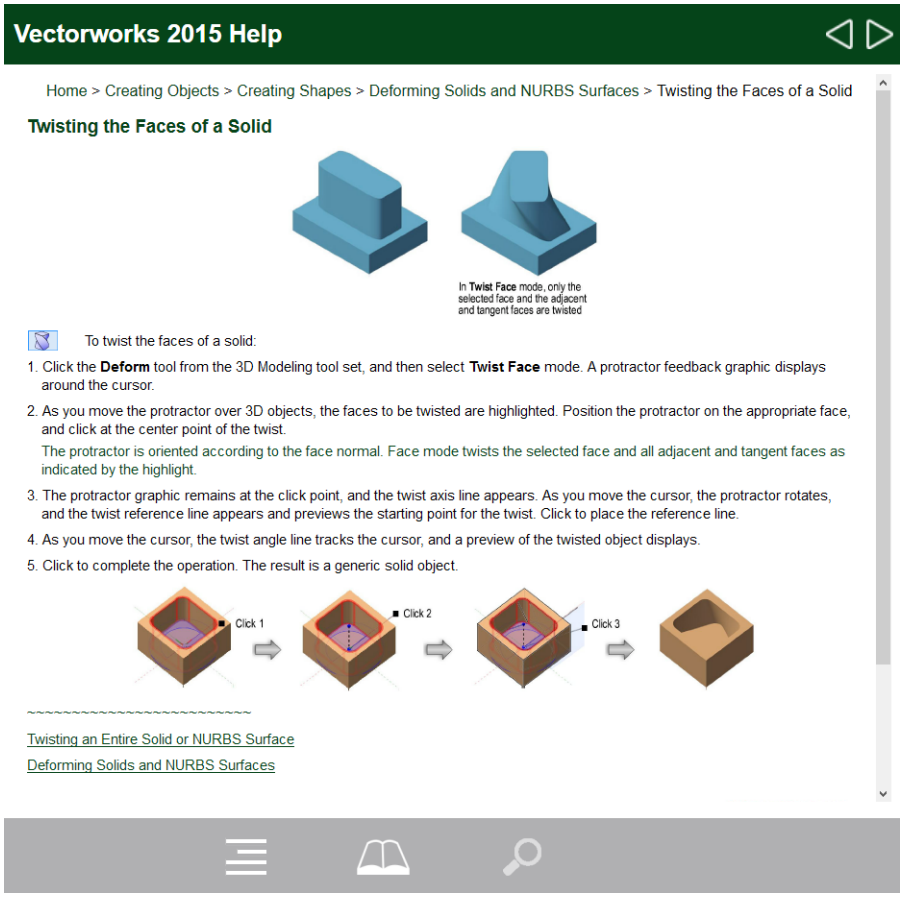
On a typical desktop screen, the help window is divided into two resizable sections containing the topics and a Navigation pane, containing the Table of Contents and a Glossary. If you do not see the Navigation pane, click the divider arrow to open the pane on your desktop, or reposition the adjustable divider between the pane and the topic panel to view more or less of the topic. Click the vertical divider to toggle the Navigation pane display.



From single-topic view on the desktop browser, click the divider arrow to open the Navigation pane, or reposition the divider.

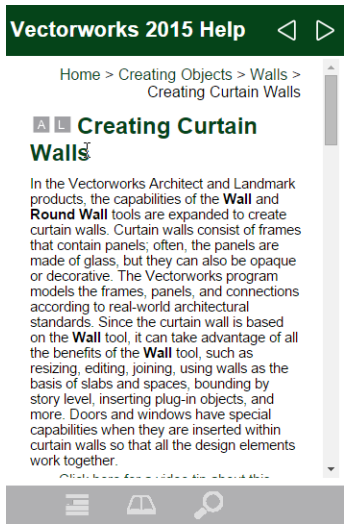
Tablet View

On a smaller screen, or on a tablet, the help system displays the topic, with buttons to the right or bottom depending on orientation, for accessing the table of contents, glossary, and search. Click a button to access one of the functions. There are no **Print** or **Back to Top** buttons in tablet view.



Phone View

On a much smaller screen, such as a cell phone, the three buttons are located at the bottom of the screen, to maximize the topic viewing area. There are no **Print** or **Back to Top** buttons in phone view.



Context-sensitive Help

Context-sensitive help is available from within the Vectorworks program. If you need information about a particular tool or command, select **Help > What's This?** and move the question mark cursor to the tool or command of interest, and then click it.

The "What's This" question mark cursor can be deactivated by selecting **Help > What's This** once again.

If you need information about a particular dialog box or a selected tool, press the keys listed in the table.

Platform /OS	Help key
Windows	F1
Mac (pre-Leopard)	Help key, Cmd + ?, F1
Mac (Leopard and later)	Help key, F1
Mac (aluminum keyboard)	Fn + F1

Vectorworks cannot directly access context-sensitive help for some context menu commands or the Worksheet menu commands. To obtain help on these items, launch the help and locate the help topic by conducting a help search.

The help you need opens in a single-topic window. You can simultaneously view the help topic and the Vectorworks application. Since it is assumed that you have a dialog box open in Vectorworks when seeking help, the same default dialog box generally does not also display in the help.

Help System Updates

If significant changes occur to the Vectorworks application or the help information during the Vectorworks product cycle, Nemetschek Vectorworks may release an update to the help system. This update is automatically available online. The local help is also updated automatically when a Vectorworks software service pack is updated or installed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Help System Features
Conventions

Help System Features

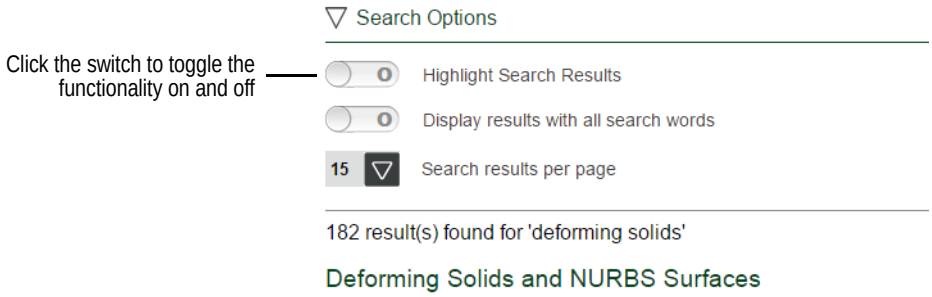
Searching the Topics

Enter text in the Search box and press Enter, or click the search loupe, to search the help system.

While searching, the progress is displayed under the Search box. When the search is complete, the number of results found displays. If no results are found, "No results found" displays.

If the search is progressing slowly or appears frozen at 0%, connection or bandwidth issues may be impeding the search progress. The Session tab of the Vectorworks preferences allows switching to the local version of the help in this type of situation.

The first toggle switch under **Search Options** turns highlighting on or off. The second switch enables "AND" search functionality if desired. Turn the toggle off to find results that contain one or more of the search terms ("OR" functionality).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



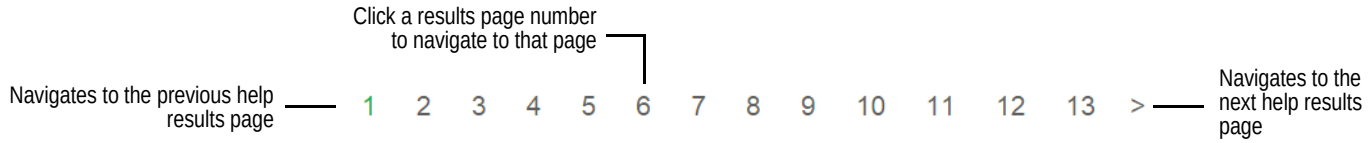
When entering search text:

- Enter one or more words and toggle the **Show results that include all search words** option on to search the help for topics that contain all the words, or turn it off to search the help for topics that contain any of the words
- Enter text within quotation marks to search the help for a text string

For example, searching for the words *color palette* finds topics that contain both the word color and palette. However, searching for the string "color palette" (with quotation marks) finds topics that contain the term color palette.

The search results list the topics that contain the search text or string. Topics are listed in order of relevance, and display both the topic heading and an excerpt from the text for you to determine the topic's relevance.

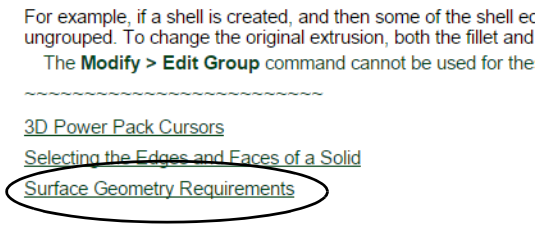
If there are more search results than can be displayed on one page, buttons at the bottom of the page allow navigation among search result pages.



Click a search result title to open the associated topic. The search term(s) are highlighted within the topic, if highlighting is enabled by the toggle switch.

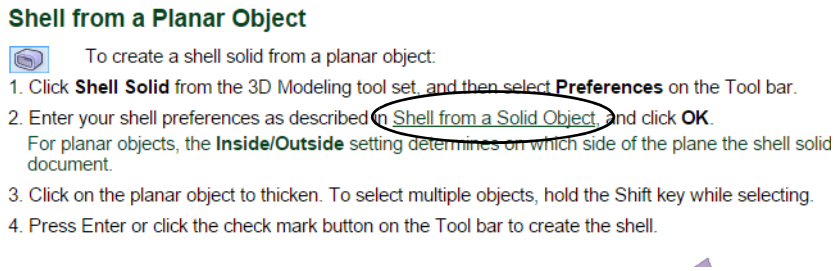
Related Topics

At the end of certain topics, a list of additional topics with information related to the current topic is displayed. Click one of the topics to view the additional information. To return to the original topic, use the browser's Back button.



Cross-references

Cross-references located within a topic are underlined and in color. Click these links to navigate to more information about the underlined topic. To return to the original topic, click the browser's **Back** button.



Browse Navigation

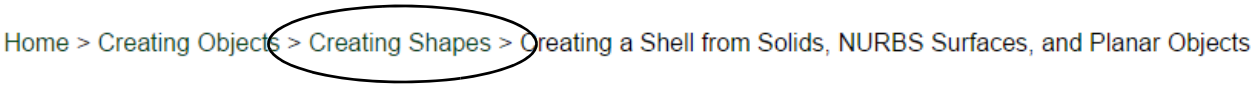
The browse sequence buttons allow you to scroll through the help topics, in order. These special navigation buttons display at the top of the topic window (the buttons also display at the bottom of the topic window for tablet or phone view). Click **Previous** and **Next** to navigate through the topics.

Desktop view browse buttons

Phone view browse buttons

Breadcrumb Navigation

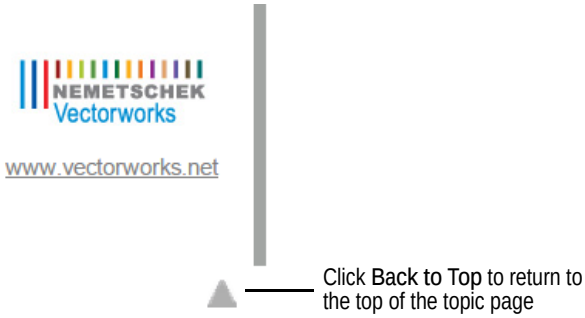
At the top of each topic page, breadcrumb navigation displays the current topic path and facilitates navigation to other topics along the path. Click any part of the breadcrumb path to navigate to that topic.



Back to Top

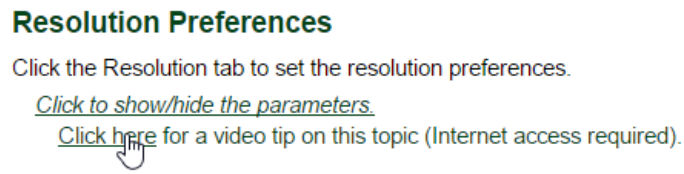
In desktop view, longer topics include a **Back to Top** button at the bottom of the page. Click the button to return to the top of the topic.

Tablet and phone view do not include Back to Top functionality.



Movie Links

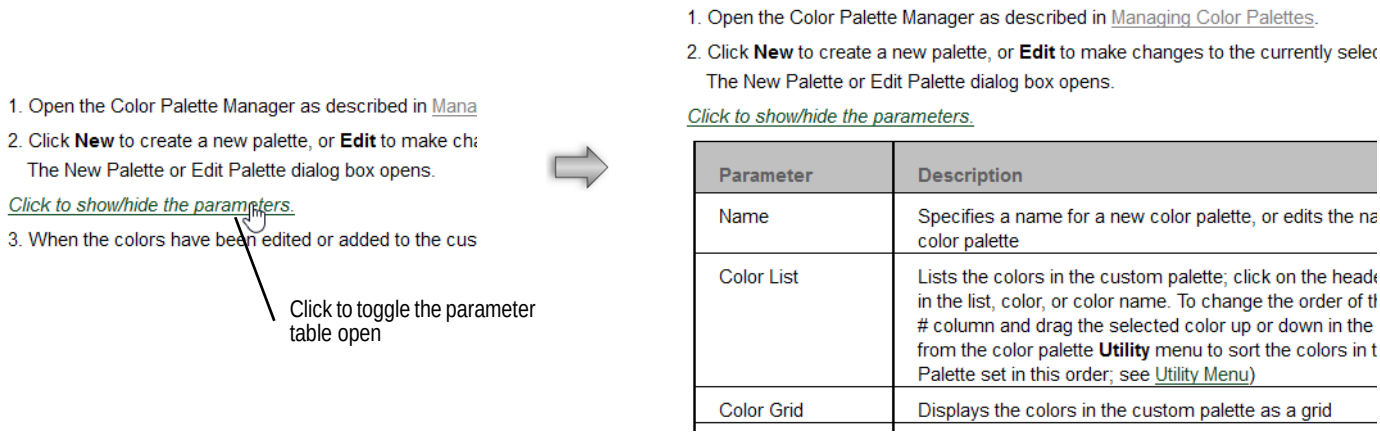
The Vectorworks YouTube channel contains video technical tips with an audio/visual explanation of various concepts. If a topic in the help has a related video tip, a link in the help directs you to the associated Vectorworks YouTube movie.



Parameter Tables

The parameters of every dialog box are described in tables. By default, these tables are hidden so that you can read the steps of a procedure easily first. To view a parameter table, click the link "Click to show/hide the parameters." This link toggles the table display on and off.

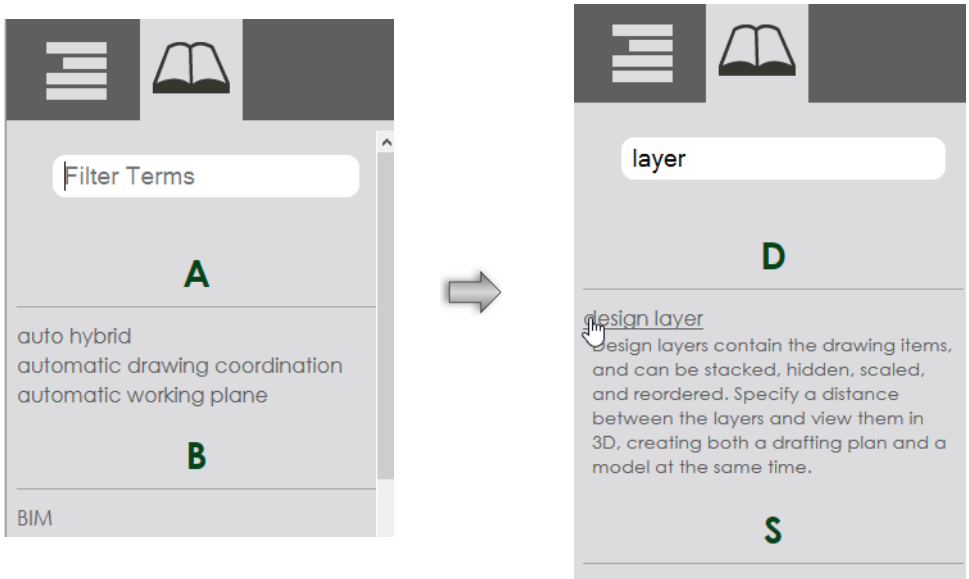
When accessing a topic from a search, the table may be showing by default, if the search term is found within the table and the **Highlight Search Results** option is turned on.



Glossary

Click the **Glossary** button from the navigation panel.

The Glossary pane displays an alphabetical listing of frequently encountered terms in the Vectorworks program. Scroll to the term of interest and click it to view its definition. The glossary definitions can be filtered by typing a term in **Filter Terms**.



Printing Topics

Click the **Print** button (desktop view only) to print the currently displayed topic through the browser's print function.

Tablet and phone view do not include print functionality.

Conventions

Vectorworks is a cross-platform product. Most Vectorworks features are the same in the Windows and Mac versions; however, in some instances, differences exist. These are explained in detail.

See the Commands and Tools PDF, available from the Help system Table of Contents, for a detailed list of the products where each tool and command is available.

The conventions are as follows:

Convention	Definition
D	Indicates functionality that exists in each of the Vectorworks Design Series products as well as Designer. As an example, if you have Spotlight, you have the functionality.
A	Indicates functionality that exists in the Vectorworks Architect and Designer products only
L	Indicates functionality that exists in the Vectorworks Landmark and Designer products only
S	Indicates functionality that exists in the Vectorworks Spotlight and Designer products only
A L	Indicates functionality that exists in the Vectorworks Architect, Landmark, and Designer products only. As an example, if you have Architect or Landmark, you have the functionality.
A S	Indicates functionality that exists in the Vectorworks Architect, Spotlight, and Designer products only
R	Indicates functionality that exists in the Renderworks product
S R	Indicates functionality that exists in the Vectorworks Spotlight and Renderworks or Designer and Renderworks products only
(Mac)	Mac-specific instruction
(Windows)	Windows-specific instruction
Enter (Windows)/return (Mac) and Backspace (Windows)/delete (Mac)	To reduce unnecessary wordiness, both operating system terms will not be used; we have standardized upon "Enter" and "Backspace." The meaning should be evident for users of either operating system.
bold text	Indicates a specific button, command, class, or explicitly named item
indented text	Indicates a note, tip, or warning
click	Click the mouse button and release. The left button is always implied in Windows.
double-click	Click two times quickly on the mouse button and release

Convention	Definition
right-click	Click with the right mouse button and release; on the Mac, hold down the Ctrl key and click the mouse
Key and mouse combination, as in Shift-click	Hold down the key(s) and click; in this example, hold down the Shift key and click
Key combination, as in Alt+Shift+D	Hold down the key(s) and press the specified letter or number key; in this example, hold down both the Alt and Shift keys, and press the D key
click-click	Click the mouse button once and release. Move the cursor to the desired location and click again. This is the default drawing preference for the program at installation. See “Edit Preferences” on page 49 to change the drawing preference to click-drag.
click-drag	Click once with the mouse button and do not release. Drag (move) the cursor to a desired location and then release.
select	Click an object with the mouse, or click-drag over an object with marquee selection, to highlight it. The object is highlighted, and/or “handles” display on the object to indicate that it is currently active. This term also refers to executing menu commands.
Context menu	A menu command accessed by a Ctrl-click (Mac) or right-click (Windows)

All instructions are based on click-click drawing.

Additional Resources

There are a variety of technical support, training, and other options available to you to ensure your success with the Vectorworks product and to support your business needs.

The **Help** menu has links to the Getting Started Guides, Vectorworks Knowledgebase Search, Vectorworks Cloud Services, and Vectorworks Service Select websites (Internet access required).

Technical Support

Technical support is available for registered users in several ways. International users should contact their local reseller for details concerning technical support; for reseller information, see www.vectorworks.net/international

United States users can contact Technical Support using the following methods:

- Call 443.542.0411
- Send a tweet to @VectorworksHelp on Twitter
- Send an email to tech@vectorworks.net
- Visit the technical support knowledgebase at kbase.vectorworks.net

When you contact Technical Support, please have access to your computer and be ready to tell the specialist:

- Vectorworks software serial number
- Vectorworks software version number
- Operating system
- Type of computer being used
- Amount of RAM installed in the computer
- List of any recent changes to the computer’s setup (such as new fonts, software, or hardware)
- A brief description of the problem with specific details about what actions were taken prior to the problem’s occurrence. The more information you can give your support specialist, the easier it will be to solve your problem quickly.

Troubleshooting

Troubleshooting a problem prior to calling Technical Support will also aid in a speedy resolution. Basic troubleshooting tips include:

- Test to see if the problem occurs in a new, blank file
- Test to see if the problem occurs system wide (especially print and font problems)
- Copy and paste part of the document to a new file to see if the problem persists
- Run the computer in Safe Mode (Windows) to see if there is a system conflict
- Check the community message board to see if the problem has already been reported or resolved - techboard.vectorworks.net

Training

For details about the following training options, visit www.vectorworks.net/training

- Tutorial manuals
- One-on one online training
- Customized onsite training
- Hands-on classroom training

To contact a training specialist, call 1.877.202.8871 (in the United States) or send an email to training@vectorworks.net

Other Resources

The following additional resources are available to Nemetschek Vectorworks users:

- Vectorworks community board - techboard.vectorworks.net
- Vectorworks YouTube channel - www.youtube.com/vectorworks
- Independent local user groups - www.vectorworks.net/community/usergroups.php
- Planet Vectorworks, our company news site - planet.vectorworks.net
- Various social media sites, such as Facebook, Twitter, Delicious, LinkedIn, and Flickr
- LISTSERV® user lists - www.vectorworks.net/community/mailinglists.php

Vectorworks Service Select

Vectorworks Service Select is a software service agreement that allows subscribers to purchase Vectorworks software at a fraction of the cost of a new license. Subscribers also get exclusive content and learning resources and always have access to the latest Vectorworks products. To learn more about Vectorworks Service Select, visit http://serviceselect.vectorworks.net/learn-more

Vectorworks Cloud Services

Share files and make design decisions from any location with Vectorworks Cloud Services, a member benefit for Vectorworks Service Select subscribers. Accessibility options, including a desktop application and the Vectorworks Nomad iOS™, Android™, and Kindle Fire™ apps, allow you to view, mark up, share, and synchronize Vectorworks files across devices. Learn more about Vectorworks Cloud Services at http://www.vectorworks.net/cloudservices and check its availability in your location by visiting: http://serviceselect.vectorworks.net/learn-more

Vectorworks Nomad iOS has the additional benefit of being able to view and present your 3D drawings rendered in OpenGL on your iOS mobile device.

Vectorworks Remote App

The Vectorworks Remote app connects your mobile device to your Vectorworks desktop using a remote connection plug-in. Visualization remotes are available for Renderworks and navigation remotes are available for Vectorworks

Design Series. Interactively use the remotes to view, navigate, and present designs without having to be at your desktop. To learn more, visit http://www.vectorworks.net

Knowledgebase

One of the primary technical support resources available to you is our Knowledgebase. This resource provides answers to the most common technical questions, as well as video tutorials and tech tips, all in a central location. Access our Knowledgebase at http://kbase.vectorworks.net

Getting Started Guides

These videos and printed guides provide you with a strong base as you learn to use Vectorworks software to design, communicate, and document your ideas. View an introduction and learn more about these resources by visiting: http://www.vectorworks.net/training/getting-started-guides

Copyright Information

Vectorworks Documentation

The Vectorworks User's Guides were written and illustrated by Alexandra Duffy, Teresa Heaps, Kristin Bailey, Martin Brown, and Susan Collins.

The Kohler plumbing fixture libraries supplied with certain Vectorworks Design Series products are based on geometry from the Kohler Company.

The Marvin door and window libraries supplied with certain Vectorworks Design Series products are based on geometry from the Marvin Company.

The Loewen door and window libraries supplied with certain Vectorworks Design Series products are based on geometry from the Loewen Company.

The Weather Shield® door and window libraries supplied with certain Vectorworks Design Series products are based on geometry from Weather Shield Mfg. Inc.

The AGA/Rayburn appliance library supplied with certain Vectorworks Design Series products are based on geometry from AGA Foodservices Group Plc.

The Copper Beech Millwork molding profiles supplied with certain Vectorworks Design Series products are based on geometry from Copper Beech Millwork, an Amherst Woodworking and Supply, Inc., Company.

The Sub-Zero and Wolf appliance libraries supplied with certain Vectorworks Design Series products are based on geometry from the Sub-Zero Freezer Company, Inc., and the Wolf Appliance Company, Inc.

The Knoll furniture library supplied with certain Vectorworks Design Series products is based on geometry supplied by the Knoll Furniture company. For more information on their products, see www.knoll.com

The Herman Miller (HMI) furniture libraries supplied with certain Vectorworks Design Series products are based on geometry from Herman Miller, Inc.

The Unilock® hatch library supplied with certain Vectorworks Design Series products is based on geometry from Unilock.

The Benjamin Moore® paint color palettes supplied with certain Vectorworks Design Series products are based on color values supplied by Benjamin Moore & Co.

The Sherwin-Williams paint color palettes supplied with certain Vectorworks Design Series products are based on color values supplied by The Sherwin-Williams Company.

The Pittsburgh® Paints paint color palette supplied with certain Vectorworks Design Series products is based on color values supplied by PPG Industries.

The Landmark plant database was created using public domain data. Nemetschek Vectorworks assumes no responsibility for inaccuracies, omissions, or errors in this data.

Lightwright is a trademark of John McKernon.

High End Systems' lighting symbols provided by, and used with the permission of, High End Systems, Inc. High End Systems is a registered servicemark of High End Systems, Inc., a division of Barco N.V.

Clay Paky lighting symbols are based on geometry provided by Clay Paky S.P.A.

Coemar lighting symbols are based on geometry provided by Coemar manufacturing.

Martin® lighting symbols are based on geometry provided by Martin Professionals A/S.

Selecon lighting symbols are based on geometry provided by Selecon, a division of Philips Global Entertainment Lighting Group.

Vari-Lite® lighting symbols are based on geometry provided by Vari-Lite Corporation, a division of Philips Global Entertainment Lighting Group.

Color Kinetics™ lighting symbols are based on geometry provided by Color Kinetics™, a division of Philips Solid State Lighting Solutions.

James Thomas Lighting Symbols and Truss Symbols are based on geometry provided by James Thomas Engineering™, Ltd.

ADB lighting symbols are based on geometry provided by ADB-TTV Lighting Technologies.

DTS lighting symbols are based on geometry provided by DTS, The Lighting Company.

JB lighting symbols are based on geometry provided by JB Lighting.

Eurotruss truss symbols are based on geometry provided by Eurotruss.

Atlas Sound® speaker symbols are based on geometry provided by Atlas Sound Lp.

EAW Sound speaker symbols are based on geometry provided by EAW sound, a division of LOUD Technologies Inc.

d&b audiotechnik® speaker symbols are based on geometry provided by d&b audiotechnik GmbH.

Meyer Sound® speaker symbols are based on geometry provided by Meyer Sound Laboratories, Inc.

Rosco gobo textures and color information provided by and used with the permission of Rosco Laboratories, Inc. Rosco is a registered trademark of Rosco Laboratories, Inc.

Apollo gobo textures and color information provided by and used with the permission of Apollo Design Technology, Inc. Apollo is a registered trademark of Apollo Design Technology, Inc.

GAM gobo textures and color information provided by and used with the permission of GAM the Great American Market, Inc. GAM is a registered trademark of GAM, Inc.

GOBOLAND gobo textures provided by and used with permission of GOBOLAND. GOBOLAND is a registered trademark.

GoboMan gobo textures and color information provided by, and used with the permission of, GoboMan.

LEE gobo textures and color information provided by and used with the permission of LEE Filters USA.

Rosco, Apollo, GAM, GOBOLAND, GoboMan, and LEE images and color information are copyrighted. All rights reserved. Their resale, redistribution, or use in derivative works is prohibited.

Thomas Register is a trademark of, and PartSpec is a registered trademark of Thomas Publishing Corp.

All other brand or product names are trademarks or registered trademarks of their respective companies or organizations.

For Defense Agencies: Restricted Rights Legend. Use, reproduction, or disclosure is subject to restrictions set forth in subparagraph (c)(1)(ii) of the Rights of Technical Data and Computer Software clause at 252.227-7013.

For civilian agencies: Restricted Rights Legend. Use, reproduction, or disclosure is subject to restrictions set forth in subparagraphs (a) through (d) of the commercial Computer Software Restricted Rights clause at 52.227-19. Unpublished rights reserved under the copyright laws of the United States. The contractor/manufacturer is Nemetschek Vectorworks, Incorporated, 7150 Riverwood Drive, Columbia, MD, 21046, USA.

Contributions

Contributions were provided by Biplab Sarkar, Robert Anderson, Kevin Linzey, Tom Urie, Dave Donley, Mark Flamer, Wes Gardner, Eric Gilbey, Gunther Miller, Andrew Bell, Don Ward, and Tim Connors.

Riccardo Martire's MasterWorks contest file submission was used as the basis for a variety of illustrations. Evanston Green Townhomes file used as the basis for illustrations © 2008 Nathan Kipnis Architects, Inc. Special thanks to Norm Harris (Marvin Windows and Doors).

Registration and Updates

If you have not already done so, please register your copy of the Vectorworks software with Nemetschek Vectorworks at http://register.nemetschek.net.

If you would like to receive automatic notification of Vectorworks software updates, you can select to automatically check for updates on a weekly or monthly basis from the Session tab of Vectorworks preferences (Session Preferences).

New Features

The following table contains a list of new and improved features for the Service Pack 3 release of Vectorworks 2015 software, and indicates the section where the functionality is documented.

Feature	Purpose	Location	Product
BIM Collaboration Format (BCF) file viewer	The **Open BCF Viewer** command allows you to open BCF files produced by other BIM applications, to view issues, proposals, and change requests for the project	"Viewing BCF Files" on page 1759	Architect and Landmark
IFC export options for walls and slabs	• The Simplified Geometry option for the **Model View** merges components, and exports a simple extrude for each wall and slab object • The **Export Walls/Slabs by components** option exports each component of walls and slabs as sub-elements	"Exporting IFC Projects" on page 1755	Architect and Landmark

The following table contains a list of new and improved features for the Service Pack 2 release of Vectorworks 2015 software, and indicates the section where the functionality is documented.

Feature	Purpose	Location	Product
Improvements to the Lightwright Data Exchange	Improvements to coordinate with a future release of Lightwright include: • a new **Address** parameter for lighting device objects, to be used in automatic universe assignment • a new option to exchange inventory lists between Vectorworks Spotlight and Lightwright • the ability to receive updated lighting device rotation information from Lightwright	"Lighting Instrument Properties" on page 877; "Universe Assignment in Vectorworks Spotlight" on page 99; and "Importing and Exporting Vectorworks Spotlight Data" on page 1703	Spotlight

The following table contains a list of new and improved features for the initial release of the Vectorworks 2015 software, and indicates the section where the functionality is documented and the product to which the feature applies.

New Features videos are available from the website.

Feature	Purpose	Location	Product
Vectorworks help system	The Vectorworks help system, keyboard shortcut file, and list of tools and commands have been updated to reflect version 2015 functionality	Not applicable	
Cocoa® API	Replaces Apple's Carbon® API with their modern Cocoa API	Not Applicable	Fundamentals

Feature	Purpose	Location	Product
Closing 2D polygon, polyline, polygonal wall network, or path-based objects	After completing all but the final click, press the keyboard shortcut (K by default) to automatically close polygon, polyline, polygonal wall network, or path-based objects	"Miscellaneous Keys" on page 3	Fundamentals
Unified install	Vectorworks no longer requires reinstalling when upgrading products or upgrading from an evaluation version	"Installing Vectorworks Products" on page 21	Fundamentals
Default workspaces for student and evaluation versions of Vectorworks software	Installation of student and evaluation versions of the software now includes a question that determines the most relevant default workspace to open on product launch	"Installing Vectorworks Products" on page 21	Fundamentals
High-resolution display	The Vectorworks application now supports high-resolution display devices	"Screen Resolution" on page 25	Fundamentals
Mac Retina display support	All icons and static images in the Mac version of the software have been recreated to display appropriately on Retina as well as standard resolution devices. The Workspace Editor and Plug-in Manager allow users to add high-resolution images in addition to standard-resolution images.	"Screen Resolution" on page 25; "Modifying Tool Palettes and Tool Sets" on page 1840; and "Plug-in Definition" on page 1785	Fundamentals
Description field for layers and classes	The edit dialog boxes for design layers, sheet layers, and classes now each have a **Description** button, which allows you to store additional information. Descriptions display as screen tips on the layer and class lists on the Organization dialog box, Navigation palette, and (on Mac only) View bar. Descriptions also display on the New Design Layer, New Sheet Layer, and New Class dialog boxes, so that you can see descriptions of layers and classes before you import them.	"The View Bar" on page 37, "Setting Design Layer Properties" on page 165, "Setting Sheet Layer Properties" on page 170, "Setting Class Properties" on page 179, and "The Navigation Palette" on page 199	Fundamentals
Updates for **Center on objects after view change** preference	Helps you feel oriented when switching views or changing layers	"Vectorworks Display Preferences" on page 50	Fundamentals

Feature	Purpose	Location	Product
The Quartz imaging display technology is now always used on Mac systems	As part of the 64-bit application change, Apple's 32-bit QuickDraw technology can no longer be used. This caused the following additional changes: • In the Vectorworks Display preferences, the Quartz imaging option was removed. • On the Edit Design Layers dialog box, the **Transfer Mode** option for controlling layer display is no longer available. • The **Export PDF (Quartz only)** command was removed; Vectorworks Fundamentals users can print to PDF using the operating system print option.	"Vectorworks Display Preferences" on page 50, "Setting Design Layer Properties" on page 165, and "Setting the Design Layer Transfer Mode" on page 168	Fundamentals
Ability to disable class hierarchical display in pop-up menus	You can now opt to disable hierarchical display of classes in pop-up menus throughout the software, including in the Object Info palette, the View bar, and dialog boxes	"Session Preferences" on page 52	Fundamentals
3D render mode and projection preferences	You can now set the preferred 3D render mode and projection, applied whenever you switch a drawing from Top/Plan into a 3D view	"3D Preferences" on page 54	Fundamentals
Remove Obsolete OpenGL Options	Removes outdated and obsolete OpenGL options from the user interface.	"3D Preferences" on page 54 "OpenGL" on page 1590	Fundamentals
View transition animation	Adds the option to animate the transition when switching views	"Interactive Preferences" on page 56	Fundamentals
Sheet border improvements	You can now add a countersignature field to the sheet border, as required by the regulations of certain countries. You can now adjust sheet border marker line length and weight and grid line extensions.	"Specifying Additional Sheet Border Settings" on page 81	Fundamentals
Beam angle, field angle, and centerline class assignment preference	The beam angle, field angle, and centerline of lighting instruments can be shown or hidden, and each can be assigned to individual classes for better control of the beam appearance, especially in viewports	"Lighting Device Setup" on page 95	Spotlight
Add custom pop-up parameter lists for lighting devices	You can now create a pop-up list of pre-defined parameter options for lighting devices	"Specifying Lighting Device Parameter Display" on page 98	Spotlight

Feature	Purpose	Location	Product
Page boundary SmartCursor cues	Improved SmartCursor cues assist users in locating the edges of the page boundary	"SmartCursor Cues" on page 149	Fundamentals
Screen tips for referenced design layers	For referenced design layers, the full layer name and the source file name display as a screen tip when the cursor is positioned over the italicized layer name in the Organization dialog box, Navigation palette, and (for Mac only) the Layers list on the View bar	"The Organization Dialog Box" on page 155, "The Navigation Palette" on page 199, and "The View Bar" on page 37	Fundamentals
Simplified story workflows	It is now much easier to create and manage stories. Story levels do not need to be associated with a layer, if the purpose of the level is to constrain objects. This reduces the number of design layers required to create stories.	"Setting Up the Building Structure with Stories" on page 172, "Creating Layers" on page 162	Architect
Text style as a class attribute	The Edit Class(es) dialog box now has a **Use Text Style at Creation** option, which allows you to assign a text style to a class, as well as specify that the style be applied when an object in that class is created. The **Text Style** field is now an option on the Object Info palette for several plug-in objects that have embedded text, such as drawing labels and callouts. Select <Class Text Style> to apply the class style to the object.	"Setting Class Properties" on page 179 and "Applying Text Styles" on page 394	Fundamentals
Visibility tool improvements in viewports	You can now use the **Visibility** tool to change class and layer visibilities in viewports	"The Visibility Tool" on page 195	Fundamentals
Assign objects to classes and layers in the Navigation palette	A new **Assign to Selection** context menu command in the Navigation palette provides an easy alternative method for assigning objects to classes and design layers	"The Navigation Palette" on page 199 and "Assigning Objects to Classes and Layers" on page 998	Architect, Landmark, Spotlight
Symbol editing shortcuts for the Resource Browser	Adds editing modes directly to the context menu in the Resource Browser	"Working with Resources" on page 225 "Editing Symbol Definitions" on page 247	Fundamentals
Locate the repetitive unit symbol in the Resource Browser	The **Locate Symbol in Resource Browser** context menu command is now available for repetitive unit objects	"Finding Resources" on page 232	Architect, Landmark

Feature	Purpose	Location	Product
Show and reverse direction for lines, polylines, and 2D/3D polygons	Show the direction for lines, polylines, and 2D/3D polygons; reverse direction as necessary for consistent operations and appearance	"Displaying and Reversing Object Direction" on page 326	Fundamentals
Deform Tool	The **Deform** tool (formerly the **Twist** tool) twists, tapers, bulges, and bends solid objects, offering expanded modeling capabilities to designers	"Deforming Solids and NURBS Surfaces" on page 361	Fundamentals
Variable spacing for tab stops in text objects	You can now add multiple tab stops to text objects and control the spacing of each one independently	"Adding Tabs to Text" on page 386	Fundamentals
Tracking support for text objects	For text objects, the new **Tracking** field on the Object Info Palette controls the horizontal spacing between characters	"Changing Other Text Features" on page 391	Fundamentals
Space object gross boundary definitions	The gross boundary definition of space objects can now be set to the outer wall core or center wall core, to accommodate a variety of standards and requirements	"Space Settings: 2D Boundaries & Area Pane" on page 410	Architect
Multiple labels for space object	Individually format and position as many as three space labels per space object	"Space Settings: Space Label 1, 2, and 3 Panes" on page 412	Architect
Improved control of leader lines for space objects	You now have greater control over the placement of leader lines for space labels	"Space Settings: Leader Line Pane" on page 413	Architect
Roof framing improvements	• Increases functionality of the **Roof Framer** command • Increases accuracy of 3D roof framing models • Provides new option for specifying how the end of a rafter meets the plate • The message bar now displays the progress of the roof framing process	"Framing a Roof" on page 461	Architect
Framing Member tool enhancements	Framing member objects now have a **Length** parameter and a directional indicator	"Creating Framing Members" on page 471	Architect
Joist from poly progress indicator	The message bar now displays the progress of the joist creation process	"Creating Framing Members" on page 471	Architect

Feature	Purpose	Location	Product
Curtain walls	A new feature of the **Wall** and **Round Wall** tools is the ability to create curtain walls. A new **Edit Curtain Wall** tool edits the frames and panels that make up the curtain wall. Curtain wall styles are an extension of wall style resources. New worksheet functions have been added for curtain walls. Doors and windows have been improved to automatically size to the bounding frames when placed as a special curtain wall door or window.	"Creating Curtain Walls" on page 522, "Editing Curtain Walls" on page 551, "Inserting Windows in Vectorworks Design Series" on page 594, "Inserting Doors in Vectorworks Design Series" on page 617, "Worksheet Functions" on page 1344	Architect, Landmark
Rectangle mode for **Wall** tool	The **Wall** tool now offers a **Rectangle** mode that allows quick, two-click drawing of rectangular walls. Walls drawn this way interact to speed the drawing of complex wall systems.	"Creating Walls" on page 497 and "Automatically Joining Walls in Rectangle Mode" on page 543	Fundamentals
Wall object and slab object dialog box improvements	Dialog boxes relating to wall and slab attributes and wall component and slab component attributes now offer an easy way to duplicate components, greater control of fill and pen attributes including complex line types, and relocated texture controls	"Drawing Straight Walls" on page 499, "Defining Components for New Walls" on page 514, "Creating Slab Styles" on page 486, and "Creating Slab Components" on page 491	Fundamentals
Wall component constraints	Like walls, wall components can now be individually constrained at their top and bottom to layer elevation or layer wall height. Vectorworks Architect users also have the option to constrain to story levels, so that the components automatically adjust when the story elevations change.	"Creating Wall Components" on page 513	Architect
Class support for door and window 2D geometry	All attributes of door and window 2D geometry can now optionally be controlled by class	"Door Settings: 2D Visualization Pane" on page 619, "Window Settings: 2D Visualization Pane" on page 596 and various locations	Fundamentals

Feature	Purpose	Location	Product
Centerline Marker for Doors and Windows	The **Door** tool and **Window** tool now offer a centerline marker option	"Door Settings: Centerline Marker Pane" on page 627 and "Window Settings: Centerline Marker Pane" on page 603	Architect, Landmark
New Stair configurations	The **Stair** tool adds two stair configurations (Double-U Stair, Double Landings and Triple-L Stair, Triple Landings) that support the design of O-shaped stairs. The new stair configurations and angled landings capability (listed separately) eliminate the need for a separate **Custom Stair** tool, which has been removed from all workspaces; the **Stair** tool has been added to the Landmark workspace.	"Stair Settings: General Tab" on page 636	Architect, Landmark
Angled landings for stairs	The **Stair** tool now supports angles other than 90 degrees for the landings of some L- and U-shaped stair configurations	"Stair Settings: Geometry Tab" on page 638	Architect, Landmark
Toilet stall improvements	Toilet stall doors can be set to be open or closed in 3D view	"Inserting Toilet Stalls" on page 669	Architect
Variable batter slopes for **Create Pad from Grade Limits** command	When creating a pad automatically from grade limits, you can now set individual slope values for each side of the pad; in addition, rise-over-run, angle, or percent values can be used to specify slope values	"Creating a Pad from Grade Limits" on page 713	Landmark
New Roadway Polyline object	A new **Roadway (Poly)** tool makes it much easier to create and edit road segments	"Creating Polyline Roadways" on page 722	Architect, Landmark
New Roadway Custom Curb object	A new tool joins roadway sections by creating a free-form custom curb roadway, for intersections, exits, dead-ends, and more	"Joining Roadways with a Custom Curb" on page 724	Architect, Landmark
Roadway object improvements	Improves the consistency between roadway objects for increased ease of use: • Allows control of paving and curb appearance and/or visibility by class • Provides an arc center locus for tee and curved roads • Measures the radius to the center of tee and curved roads	"Creating and Joining Straight and Curved Roadways" on page 733	Architect, Landmark
On-screen indicators for site model errors	Grade modifier errors that prevent a site model from updating correctly are indicated by an icon at the site of each error	"Correcting Site Modifier Errors" on page 754	Architect, Landmark

Feature	Purpose	Location	Product
Preserve the grade of a reshaped grade object	You can now opt to preserve the grades of reshaped grade objects and automatically recalculate the elevation instead	"Specifying Global Grade Object Preferences" on page 764	Landmark
Grade object length display options	The **Grade** tool's new parameters allow the optional display of the grade object's projected and/or surface length, and grade parameters can now be entered as a rise/run ratio	"Inserting Grade Objects" on page 765	Landmark
Automatic generation of minimum and maximum values for graticule object creation	The **Graticule** tool automatically calculates the graticule's minimum and maximum latitude and longitude based on the geometry and location of the defining polyline	"Creating a Graticule" on page 780	Landmark
Plant object tight outline	A new tight **Outline** option for plant objects offers a more defined look than the existing sketch styles	"Plant Definition: Render Pane" on page 789	Landmark
Improved access to plant content libraries	It is easier to add a wide variety of plants to drawings, with plant content libraries accessible directly from the Plant Settings dialog box, Copy From Symbol dialog box, and Edit Plant Information dialog box	"Plant Settings: Definition Pane" on page 795; "Creating Plant Geometry from a Symbol" on page 791; and "Creating Landscape Areas" on page 834	Landmark
Extended customization for plant tags	You can now create multiple lines of data for the top, center, and bottom of the custom plant tag and can now edit the center and bottom tags directly in the Object Info palette	"Creating a Custom Plant Tag" on page 798	Landmark
Existing Tree tool enhancements	The **Existing Tree** tool is now even more flexible. You can specify irregular canopy shapes, and use trees with multiple trunks in calculations. Textures can be included for enhancing the 3D appearance of the trees. Additional options have been added for tree protection zone display and for the use of custom and specialized fields in the Object Info palette.	"Documenting Existing Trees" on page 818	Landmark
Landscape Area tool improvements	The landscape area is now calculated based on the site model if one exists, and displays in the Landscape Area Settings dialog box	"Creating Landscape Areas" on page 834	Landmark
Custom landscape area tag improvements	Improves the functionality of the custom landscape area tag	"Creating a Custom Landscape Area Tag" on page 838	Landmark

Feature	Purpose	Location	Product
Parking spaces terminology improvement	Handicapped parking space options have been renamed "Accessible" in keeping with current usage	"Creating Parking Spaces" on page 845	Architect, Landmark, Spotlight
Spotlight user interface improvements	The names of several Vectorworks Spotlight tools, commands, dialog boxes, and parameters have been changed to be more complete and/or consistent	Various locations	Spotlight
Option to embed the geometry in a lighting position object	When you use the **Convert to Light Position** command to convert geometry into a lighting position object, you are now prompted to either use the geometry directly in the object, or create a symbol. If you use the geometry in the object, a new **Edit Position Geometry** button will be available on the Object Info palette, to allow easier editing.	"Creating a Lighting Position Object" on page 865	Spotlight
Flip label legend settings	Adds settings to adjust orientation of the label legend for lighting instruments	"Lighting Instrument Properties" on page 877 "Spotlight Setup" on page 95	Spotlight
Soft Goods improvements	The parts of a soft goods curtain or border can now be classed individually, offering more control over appearance and visibility	"Inserting Soft Goods" on page 904	Spotlight
Video screen improvements	Improvements to all the video screen objects allow greater control over classing and the addition of a note to the label; new options for screen borders, mounting, and additional screen parameters and aspect ratio options are available to applicable video screen objects	"Inserting Video Screen Objects" on page 913	Spotlight
Audio object improvements	Speakers and speaker arrays have been enhanced with additional orientation and mounting options and the ability to specify textures for the various parts of the speaker	"Inserting Speakers and Speaker Arrays" on page 930	Spotlight
Stage Lift tool	A new tool inserts a stage lift object, an adjustable supporting device for stage equipment	"Inserting a Stage Lift" on page 940	Spotlight
Stage objects	Stage and theatrical structures can now be included in entertainment industry drawings with several new tools for stages, step, and ramp structures. The improved stage and steps are now also used by event design commands.	"Inserting Stage Structures" on page 942, "Event Design" on page 983	Spotlight

Feature	Purpose	Location	Product
Instrument summary header added	A position height header can now be added to instrument summaries. When the summary is filtered by lighting position, the height is displayed in the header.	"Creating an Instrument Summary" on page 961	Spotlight
Lighting inventory improvements	Inventory reports have been consolidated. The same inventory report that is set up from the **Lighting Inventory Setup** command is now included in generated paperwork.	"Inventory Reports" on page 969	Spotlight
Replace PICT data with bitmap data	As part of the 64-bit application change, most workflows that used PICT data now use bitmap data. • The following commands were renamed, because they now create bitmap data rather than PICT data: **Paste as Bitmap** (was **Paste as Picture**) **Import Metafile as Bitmap** (was **Import Metafile as Picture**) **Import PICT as Bitmap** (was **Import PICT as Picture**) • The **Export PICT** command was removed. • The **Import PICT** command continues to convert PICT data into Vectorworks primitive objects. • The **Import EPSF** command no longer imports the PICT format previews that can be embedded in PostScript files; only TIFF format previews are imported. • The **Export EPSF** command has different options for including previews with the PostScript data. The PICT format preview option (Mac only) was removed. All previews are now TIFF format, and there are now Full color, Grayscale, and Black and white options.	"Paste as Bitmap" on page 1010, "Importing and Exporting in Metafile Format" on page 1685, "Importing in PICT Format" on page 1671, and "Importing and Exporting in EPSF Format" on page 1679	Fundamentals
Duplicate Along Path command improvement	You can now select whether to center objects along the path during a duplicate-along-path operation	"Duplicating Objects Along a Path" on page 1017	Fundamentals
Align/Distribute context menu command	The **Align/Distribute** command is now available from the context menu for ease of operation	"Aligning and Distributing Objects in the Screen Plane" on page 1032	Fundamentals

Feature	Purpose	Location	Product
Save settings for the **Extrude Along Path** command	The settings in the Extrude Along Path dialog box are now saved between Vectorworks sessions	"Extrude Along Path" on page 1040	Fundamentals
Save settings for the **Sweep** command	The settings for angles in the Create Sweep dialog box are saved between Vectorworks sessions	"Sweeping Objects" on page 1042	Fundamentals
Improvement to **Reshape** tool **Move Edges Parallel** mode	The **Reshape** tool's **Move Edges Parallel** mode now operates on vertices, in addition to midpoints	"Move Edges Parallel Mode" on page 1046	Fundamentals
Trim and join straight walls using the **Trim** tool	Straight walls are easily trimmed and joined to intersecting straight walls using the **Trim** tool	"Trim Tool" on page 1060	Fundamentals
New Fill Patterns	New fill patterns are available in the Attributes palette, including fill patterns that are required for drawing standards of some countries	"The Attributes Palette" on page 1093	Fundamentals
Viewport rendering attribute transfer	Adds an option to the **Eyedropper** tool to specifically transfer viewport rendering settings to another viewport	"Transferring Attributes" on page 1095	Fundamentals
Opacity support for gradients	Apply opacity to gradient resources to create more dynamic presentations	"Defining Gradients" on page 1115	Fundamentals
Color menu Mac improvement	Enhanced the Color Palette search function on the Mac	"Applying Colors" on page 1132	Fundamentals
Mobile 3D Viewer	Vectorworks Nomad on iOS now supports 3D viewing of OpenGL rendered models	"Vectorworks Nomad" on page 1142 "Vectorworks Cloud Services" on page x	Fundamentals
Consolidate **Projection** and **Perspective** menus	The perspective commands have been incorporated into the Projection menu for a more streamlined workflow	"Projection" on page 1142	Fundamentals
Colored caps and edges for clip cube objects	Colored caps and edges have been added to the display of solid geometry that is cut by the clip cube. This preserves the solid appearance of the clipped model, rather than showing the model as if it were constructed from a hollow shell.	"Viewing a Model with the Clip Cube" on page 1155 and "Configuring Interactive Display" on page 116	Fundamentals
Updated Working Planes palette	The Working Planes palette has a streamlined interface and eliminates redundancies	"The Working Planes Palette" on page 1176	Fundamentals
Drawing animation improvements	Drawing animation has been simplified and no longer depends on QuickTime technology	"Animating Drawings" on page 1180	Fundamentals

Feature	Purpose	Location	Product
New option for the placement of dimension values in custom dimension standards	In the Edit Custom Dimension Standard dialog box, the **Align Vertical Text Left** field was replaced with the **Text Position** field, which has the following options: • Above/Left places the text above the dimension line for non-vertical dimensions, or left of the dimension line for vertical dimensions • Above/Right places the text above the dimension line for non-vertical dimensions, or right of the dimension line for vertical dimensions • Outside places the text outside the area enclosed by the two witness lines and dimension line, regardless of the dimension orientation	"Creating a Custom Dimension Standard" on page 1188	Fundamentals
Notes for dimensions	A **Note** field was added to the Object Info palette; the note displays on the opposite side of the dimension line from the dimension value	"Editing Dimension Properties" on page 1211	Fundamentals
Improvements for editing and removing object constraints	Adds a context menu command for quickly removing parametric constraints from an object. Also, the **Edit Constraints** command now works with multiple objects selected.	"Editing Parametric Constraints" on page 1237	Fundamentals
Improvements to the **Drawing Label** tool	Adds features to alter both the style and positioning of the drawing title and scale display	"Creating Drawing Labels" on page 1244	Fundamentals
Elevation benchmark improvements	The **Elevation Benchmark** tool's parameters offer greater customization, including new options for placing the text relative to the marker and line and new marker fill and crosshair options for ISO style benchmarks. The **Align/Distribute Leader Lines** command now applies to elevation benchmark objects	"Elevation Benchmark" on page 1251 and "Aligning and Distributing Leader Lines" on page 1035	Architect, Landmark
New **Symmetry Label** tool	Use the new **Symmetry Label** tool to easily designate the symmetry centerline as defined by some nations' drawing standards	"Inserting a Symmetry Label" on page 1257	Fundamentals

Feature	Purpose	Location	Product
New vertical text position and bubble style for callouts and keynote legends	The new Above Shoulder **Vertical Position** option and new Split Circle **Bubble Style** provide additional ways to customize callouts and keynote legends	"Inserting Callouts or Keynotes" on page 1293 and "Editing Keynotes and the Keynote Legend" on page 1298	Fundamentals
Notes Manager horizontal text alignment	New controls allow left, center, right, or justified alignment of text in callouts, keynote legends, and general notes	"Inserting Callouts or Keynotes" on page 1293, "Editing the Keynote Legend" on page 1298, and "Placing General Notes" on page 1301	Fundamentals
Callout bubble shadow improvements	Callout bubble shadows can now be drawn in custom colors	"Specifying Callout or Keynote Legend Bubble Shadow Settings" on page 1297	Fundamentals
Move default Notes Manager databases to content library	Notes Manager default databases are now located in the default content library in [Vectorworks]\Libraries\Defaults\Notes	"Notes Management with Databases" on page 1304	Fundamentals
Easier editing of space boundary classes and room finishes for space objects	Space object **2D/3D Boundary Class** can now be edited directly in a worksheet or in the Object Info palette. Room finishes can also be edited in a worksheet, and the Object Info palette now supports assigning room finishes for several selected space objects at the same time.	"Selecting and Editing Worksheet Cells" on page 1321, "Space Settings: Advanced Settings Pane" on page 416, and "Space Properties" on page 417	Architect
Worksheet menu bar	The worksheet window now has a menu bar and expanded context menus, to provide easier access to worksheet commands	"Worksheet Commands" on page 1324	Fundamentals
New worksheet number formats	The Format Cells dialog box has a new **Percentage** option on the Number tab, which converts decimal numbers to percentages in worksheet cells. The same option was added to the Number Format dialog box, which is used when you create or edit a record format. Also on the Format Cells dialog box, the YYYY-MM-DD format was added to the **Date** format options.	"Formatting Worksheet Cells" on page 1330 and "Creating Record Formats" on page 262	Fundamentals

Feature	Purpose	Location	Product
Smooth selected mesh objects	Smooth the appearance of selected meshes rendered in OpenGL or Renderworks for a more polished presentation	"Applying a Texture to an Object" on page 1530	Architect, Landmark, Spotlight, Renderworks
Improved rendering of imported meshes	The **Import SketchUp** and **Import 3ds** commands now preserve the mesh smooth normals and mesh texture mapping of the original files, resulting in a much smoother final rendering and greatly reduced model size	"Applying a Texture to an Object" on page 1530	Architect, Landmark, Spotlight, Renderworks
Wireframe rendering improvements	Improves efficiency and visual perception of wireframe rendering	"Rendering with Vectorworks" on page 1587	Fundamentals
OpenGL rendering crease edges	Improves consistency of edges in OpenGL rendering	"OpenGL Options" on page 1590	Renderworks
OpenGL rendering color option	Adds ability to turn colors on and off in OpenGL rendering while maintaining any reflectivity, transparency, and bump effects	"OpenGL Options" on page 1590	Renderworks
Hidden line rendering text and marker options	Allows you to hide or display text and markers in hidden line renderings	"Line Render Options" on page 1592	Fundamentals
Renderworks style artistic edges	Create new effects by applying artistic edges on top of realistic-shaded renderings	"Creating Renderworks Styles" on page 1596	Renderworks
3D hatching for hidden line rendering	Display planar hatches on 3D hidden line renderings wherever a textured surface would exist in a raster-rendering mode. Some of the most commonly used standard Renderworks textures are pre-associated with appropriate hatches for ease of use.	"Hidden Line Rendering with Surface Hatches" on page 1606	Renderworks
Detail callout improvements	Previously, detail callout objects were only displayed in the viewport or design layer where the detail viewport was created. You can now create multiple detail callout objects in other design layers or viewports, which are linked to the same detail viewport.	"Detail Callout Instances" on page 1635	Architect, Landmark, Spotlight

Feature	Purpose	Location	Product
Improved control over the dimension text position in viewports	The advanced properties of both sheet layer and design layer viewports have a new **Re-Position Dimension Text** option. The option re-positions rotated and flipped dimension text in the viewport according to the text position in the dimension standard. The text itself stays in the rotated and flipped orientation, but the position of the text box relative to the dimension line is changed. In addition, the **Adjust Flipped Text** option now only adjusts the dimension text to be readable, without moving it to be above the dimension line.	"Advanced Sheet Layer Viewport Properties" on page 1642 and "Advanced Design Layer Viewport Properties" on page 1644	Fundamentals
SketchUp and 3DS import improvements	Improvements to the SketchUp import process include a more organized user interface and options for importing materials and textures; 3DS import was also improved for consistency and to provide texture import when creating mesh objects	"Importing in SketchUp Format" on page 1669 "Importing in 3ds Format (3D only)" on page 1695	Architect, Landmark, Spotlight
Improvements to STL (Stereo lithography) export	• Renamed the **Export Stereo Lithography** command to **Export STL** to make it clearer that it produces STL format files. • Added options to export selected objects on the active layer, or export all visible objects on all layers; previously, the only option was to export all visible objects from the active layer.	"Exporting in STL Format" on page 1677	Fundamentals
Image import/export updates	Importing and exporting image files no longer relies on QuickTime technology	"Importing an Image File" on page 1680 and "Exporting an Image File" on page 1683	Fundamentals
Snapping option for PDF page objects	You can now enable or disable snapping for imported PDF files. Additionally, file size is reduced because snapping geometry is only generated when needed and it is never written to disk.	"Snapping to and Scaling Imported PDF Files" on page 1687	Architect, Landmark, Spotlight
PDF cropping	Allows you to crop PDF objects imported into the Vectorworks program	"Cropping Imported PDF Files" on page 1689	Architect, Landmark, Spotlight
PDF/A-1b format export	Files can be exported in the PDF/A-1b format	"Exporting PDF Files" on page 1690	Architect, Landmark, Spotlight

Feature	Purpose	Location	Product
Import/export of STEP files	Vectorworks now supports the import and export of STEP files, increasing interoperability	"Importing and Exporting in STEP Format" on page 1699	Fundamentals
Parasolid export version	The **Export Parasolid X_T (3D only)** command now supports Parasolid version 27	"Exporting in Parasolid X_T Format (3D only)" on page 1701	Fundamentals
New options to control dashed lines during DXF/DWG import	You can now select an option and set a minimum gap size to convert dashed lines to solid lines during import	"DXF/DWG and DWF Import Options" on page 1720	Fundamentals
New option to import DXF/DWG tables as worksheets	A new option allows you to import tables as worksheets, for easier modification. Previously, tables were always imported as sets of individual line and text objects.	"DXF/DWG and DWF Import Options" on page 1720	Fundamentals
DWG/DXF and DWF import terminology improvements	Updates in terminology to make import dialogs more clear as to which type of file is being imported	"DXF/DWG and DWF Import Options" on page 1720	Fundamentals
Use Vectorworks rounding precision for DXF/DWG export	The precision settings for length and angle are now set according to the setting in the Units dialog box during export. Previously, only the length precision was set, to a fixed value.	Not applicable	Fundamentals
Improved import of DWG/DXF leader lines	Standard leader and multileader lines now import as polygons or polylines with arrow markers, instead of importing as a collection of independent objects	Not applicable	Fundamentals

Feature	Purpose	Location	Product
Improvements related to the Vectorworks internal origin and user origin	Several improvements have been made regarding the Vectorworks internal origin and user origin. • When importing a DXF/DWG/DWF file, a new Location tab allows you to choose the best import option for importing files relative to the internal origin; the default option prevents objects from being imported far from the internal origin • A new Vectorworks preference displays the internal origin marker • Two new commands locate the internal origin and center the drawing to the internal origin • The User Origin dialog box has been renamed and improved • User origin and internal origin terminology throughout the Vectorworks program has been renamed to eliminate confusion	"Import Options: Location Tab" on page 1727, "Vectorworks Display Preferences" on page 50, and "Internal Origin and User Origin" on page 72	Fundamentals
New options for exporting SIA dimensions to DXF/DWG format	A new option allows you to export SIA dimensions as non-superscript AutoCAD dimensions that update automatically during reshape. Previously, SIA dimensions were always converted into non-dimension objects that looked exactly the same as they did in Vectorworks.	"DXF/DWG and DWF Export Options" on page 1734	Fundamentals
Filtering on IFC import	Apply a filter when importing IFC files to eliminate irrelevant objects, reduce file size, and collaborate more efficiently	"Importing IFC Files" on page 1753	Architect, Landmark
Export IFC command supports COBie/Facilities Management	The Export IFC command now supports the COBie/Facilities Management MVD to produce COBie-compatible IFC files	"Exporting IFC Projects" on page 1755	Architect, Landmark
Publish command improvements	• Allows you to have different settings for multiple items of the same file type • Remembers the last used set • Adds new folder options to the Publish dialog box • Saves publishing options with sets • Provides easier access to saved and published sets	"Batch Publishing" on page 1765	Fundamentals
Publish images	You can now batch publish images using the **Publish** command	"Batch Publishing" on page 1765	Architect, Landmark, Spotlight

Feature	Purpose	Location	Product
Simplified export of database tables	The process of exporting record formats to a database table has been simplified	"Automatically Creating a Database Table from a Record Format" on page 1813	Architect, Landmark, Spotlight
Workspace editor improvements	Enhances the workspace editor by improving the usability and consistency, making it easier to manage workspaces. You will now be notified if you attempt to create a workspace or if you import a third-party workspace containing tools or commands not available with your license.	"Managing Workspaces" on page 1835	Fundamentals
Save settings for resource name conflict	The settings in the Resource Name Conflict dialog box are now saved between Vectorworks sessions. This dialog box can display during both import and export of resources (Vectorworks Design Series required for export).	Various locations	Fundamentals
QuickTime® Removal from Windows Core	Animation and projection operations throughout the Vectorworks program no longer depend on QuickTime	Various locations	Fundamentals
Mac Resource Browser layout improvements	On Mac, the Resource Browser layout was made more consistent with other palettes and more compact to occupy less of the document window	Not applicable	Fundamentals
Modernize object editing mode display	The appearance and functionality of the object editing mode button has been improved to comply with operating system requirements	Not applicable	Fundamentals
Tablet command removal	The **Tablet** command used now obsolete technology and as a result was removed from all workspaces	Not applicable	Fundamentals

Documentation Improvements

The following table lists major improvements to the documentation for the initial release of the Vectorworks 2015 software.

Improvement	Purpose	Location
Complete re-engineering of the help system	The help system has been completely re-designed to make it easier to use. • The help format has switched to HTML5 and displays in your default browser • The help system is available online and can be viewed separately on a mobile device • The Fundamentals and Design Series manuals have been combined • Parameter tables are collapsed by default, allowing you to view the procedure only • The Table of Contents is task-oriented • Workflow topics and pages of relevant links help you get started with a procedure	Vectorworks Help system
New links to videos	Where relevant, additional links to instructional videos located on the Vectorworks channel have been added, to assist with comprehension.	Vectorworks Help system
New local help preference	A new Vectorworks session preference allows you to always use the locally-installed version of the help system	"Session Preferences" on page 52
Expanded and improved worksheet documentation	The worksheet functions table now includes examples for both database header and spreadsheet cells for many functions. In addition, a new tutorial topic provides a simple introduction to worksheet features.	"Worksheet Functions" on page 1344, and "Worksheet Tutorial: Creating a Wall Schedule" on page 1361
Updated shortcuts list and **Import PartSpec** command path	Shortcuts were removed from the Machine Design commands so they can be reassigned to more frequently used commands. The **Import PartSpec** command has been removed from the File menu and is now available only beneath the Machine Design menu heading.	"Vectorworks 2015 Keyboard Shortcuts" on page 1 and "Importing in PartSpec Format" on page 1668

Getting Started

Installing Vectorworks Products

The Vectorworks installation program lists the installation system requirements, any known installation issues, and any information that changed in the software after final online help was produced. To ensure a smooth installation, confirm that system requirements are met and take note of any issues or incompatibilities prior to installing the software.

The installation program installs the entire Vectorworks series of programs, and determines which products to allow access to based on the serial number.

The installation process includes an option to add material to your copy of Vectorworks software, including industry-specific content libraries that can be added to drawings through the Resource Browser (see “Resource Libraries” on page 219). If you opt not to download this material when you install Vectorworks, it can be added later by selecting **Help > Download Content** to launch the Vectorworks Package Manager, and clicking **Install** for the libraries you want to download.

Your software license allows for one copy to be made for backup purposes. The Vectorworks software should be backed up on a regular basis. These backups are invaluable in restoring user-customized settings that may be potentially lost by computer failure.

> Student and evaluation versions of Vectorworks software include an additional step at installation: selection of an area of design that best describes the kind of work to be done. This is so that the most relevant default workspace displays on product launch. For instance, if lighting design is selected as the primary use, the Spotlight workspace opens. The entire Design Series and Renderworks product line is available and the workspace can be changed in the **Tools > Workspaces** menu regardless of how the question is answered.

Adding and Removing Serial Numbers

During installation you are typically required to enter the serial number, which is provided with the Vectorworks software. If you are not prompted to enter the serial number during installation, you must enter it the first time the program is accessed. You may also need to update a serial number to enable more products or to switch from an evaluation version to a full version.

To add a serial number:

1. Open the Vectorworks program.

   The Serial Numbers dialog box opens the first time the program is accessed. Thereafter, it is accessed by clicking **Serial Numbers** from the Session tab of Vectorworks preferences.

2. Click **Add**.

   The Add New Serial Number dialog box opens.

3. Enter the serial number exactly as it appears on the label. The serial number is case sensitive. The letters o and i are not used in serial numbers; enter zero or one instead.

4. Click **Add**.

   The list of **Enabled Products** in the Serial Numbers dialog box displays all products enabled by this serial number. For security, serial numbers that begin with the letter A are partially hidden wherever they display in Vectorworks.

5. Only one serial number can be applied at a time. When the serial number is set, click **Done**.

To remove a serial number:

1. Click **Serial Numbers** from the Session tab of Vectorworks preferences.

   The Serial Numbers dialog box opens.

2. Select the serial number to remove, and then click **Remove**.

3. Click **Done**.

Viewing the Vectorworks License Agreement

The license agreement for the Vectorworks software can be viewed at any time.

To view the license agreement:

1. On Windows, select **Help > About Vectorworks**, or on Mac select **Vectorworks > About Vectorworks**.
2. Click **License** to display the license agreement text.
3. Click **OK** twice to return to the program.

Network Protection

When you run multiple copies of the Vectorworks program across a network, serial numbers are checked when the program is launched, and also periodically throughout the session.

If the same serial number is found to be in use, the All Serial Numbers In Use dialog box opens. From this dialog box, click **User Information** to display the User Name, if given, and the IP address of the other location. Click **Edit Numbers** to delete the existing serial number and enter a new one. Once the situation is resolved, click **Retry Numbers** to regain access to the program. If the serial number is still in use after three attempts, the program automatically shuts down.

Updating Vectorworks Products

Vectorworks software updates are called Service Packs, and are considered maintenance releases. There are several ways to obtain updates:

- Click **Check for Updates** from **Help > About Vectorworks**. Alternatively, select **Help > Check for Updates** (Windows) or **Vectorworks > Check for Updates** (Mac).

  A dialog box opens to indicate whether the software is up to date, or if an update is available. If an update is available, click **More Info** to open a web browser to display and download available updates. If it is inconvenient to update the software at this time, click **Cancel**.
- Select to **Automatically check for updates** on a weekly or monthly basis from the Session tab of Vectorworks preferences (see “Session Preferences” on page 52).
- Updates are available for download at www.vectorworks.net/downloads

Creating a New File

To create a new file:

1. Select **File > New**.

   The Create Document dialog box opens.
2. Select the type of document to create and click **OK** to open the new file.

Document Type	Description
Create blank document	Opens a new drawing file with the default program settings (attributes, scale, units, etc.)
Use document template	Opens a new drawing with settings that were saved in a template file; select the template to use (see “Creating Templates” on page 75 to learn more about templates)

Vectorworks searches your user folder, your workgroup folders, and the Vectorworks application folder for template files. If no templates are found anywhere on the system, the Create Document dialog box does not open. Instead, a blank new document is created automatically.

Opening a File

There are three ways to open a file that already exists. Up to eight files can be open at once in the program.

To open a file that already exists:

1. Select **File > Open**.

   The Open dialog box opens.
2. Select the name of the file or template to open.
3. Click **Open**.

   The program opens the last saved version of the selected drawing file.

To open a file that was recently used:

Select **File > Open Recent** and select a name from the list of the last 20 files that were opened or saved.

The file is opened; if the file is already open, it is moved to the front of the screen to become the current file.

To open a file from the operating system:

Double-click a file directly in Windows Explorer or Mac Finder.

If multiple versions of the Vectorworks program are installed, when you double-click to open a Vectorworks file from the operating system, the results may vary for Windows and Mac.

Windows

On Windows, file associations are determined in the Vectorworks File Associations dialog box.

- If Vectorworks file types have been associated with a particular copy of Vectorworks in the Vectorworks File Associations dialog box, double-clicking a file opens it in that particular version of Vectorworks.
- If the Vectorworks File Associations dialog box specifies that current Vectorworks file associations remain in place, double-clicking a file opens the file in the version of Vectorworks with which it is associated.

Mac

- If multiple versions of Vectorworks are currently open, double-clicking a file opens the file in whichever version of Vectorworks you opened first.
- If multiple versions of Vectorworks are installed but not open, double-clicking a file opens the file in the latest version of Vectorworks.

The Vectorworks program can open files that were created in MiniCAD 7 or in Vectorworks versions 8 and above. Files that were created in versions earlier than MiniCAD 7 must be converted to at least version 7, with a version of the program that is earlier than version 12. If an earlier version of the program is not available, contact Nemetschek Vectorworks Sales Support for assistance.

When a user attempts to open a file that another user already has open, an alert displays the name of the user who has the file open and where it is open; additional information about the file can be displayed, including its path, when it was opened, and how long it has been open.

Watermarked and Fingerprinted Files

Files from the Vectorworks educational version include a watermark. If this is the first time the file is being opened, a message displays, notifying you that the data in the file may only be used in other watermarked documents. This type of file is identified as watermarked in the title bar. If an educational version of the Vectorworks software is being used, a message displays to that effect, and also warns that any files created or modified in this version of the software will be watermarked. If you reference or import objects from a watermarked file, the current file is irreversibly converted to a watermarked file; and therefore prints with an educational watermark on every page.

Library files which contain product-specific content, such as a Landmark library file, include a fingerprint which identifies the Vectorworks Design Series product(s) required to be able to use the content. This type of file is identified as fingerprinted in the title bar. When a fingerprinted file is opened, a message displays, warning that the file or any files created from it can only be used with the indicated Design Series product(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mapping Missing Fonts

Mapping Missing Fonts

When you open a file that contains fonts that are not available on your system, the Font Mapping dialog box opens. Map fonts from the original file to a font available on your system, or open the document quickly by automatically mapping fonts to the default replacement font.

Click to show/hide the parameters.

Parameter	Description
Replace	Automatically populates when an original font is selected
With	Lists the default font and all available system fonts (the default font is the system font for the computer on which the program is installed)
Original Font	Lists the unavailable font used in the file
Replacement Font	Lists the font that will be used to replace the original font
New Mappings	Lists fonts that have not previously been mapped
All Mappings	Lists all fonts regardless of whether they have been previously mapped

To map missing fonts:

1. Populate the list of original fonts by selecting **New Mappings** or **All Mappings**.
2. If the default replacement font is acceptable for replacing all original fonts, click **OK** to open the document. Otherwise, proceed to Step 3.

   Font mappings can be changed at any time by clicking **Edit Font Mappings** from the Display tab of Vectorworks preferences.
3. Select an original font to map.

   The **Replace** field displays the selected original font, and the **With** field lists the default and all available fonts. Select the desired replacement font, and then select the next original font to map. Repeat the process until all desired original fonts have been mapped.
4. Click **OK** when all desired original fonts have been mapped.

   Mapped fonts are stored as a Vectorworks preference. If Vectorworks preferences are deleted, the Font Mapping dialog box re-displays for a file that may have previously had the fonts mapped.

Previewing Vectorworks Files from the Operating System

For Vectorworks files created in version 2008 or later, the Windows Explorer or Mac Quick Look plug-in provides image previews and thumbnail icon previews of files to more easily locate them (requires Windows Vista or later operating system, or Mac OS X 10.5+).

When a file is saved, an image of the file's current view is also saved. This image is then supplied to the operating system with the Vectorworks Windows Explorer or Quick Look plug-in.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Windows Explorer File Preview

The preview options and views available in Windows operating systems depend on your specific OS and settings. The information that may be available includes a thumbnail preview of the file, a larger preview of the saved file's current view, the date the file was last modified, and the Vectorworks version the file was created with, if known. When the Windows Search service is enabled, it can be used to locate text strings within the Vectorworks files.

Mac Quick Look File Preview

On the Mac, the preview image can be viewed in a number of ways:

- Select one or more documents within the Finder, and then press the Space Bar
- Select a document and then press the Quick Look button within the Finder (if your Mac settings display the Quick Look button)
- Select a document in the column view; the preview column contains the preview image
- Display the preview image in the preview section for **Get Info**

  A preview is not available for files saved with auto-save or batch convert.

Mac Quick Look Thumbnail Preview

Vectorworks documents can use either the document thumbnail icon or the standard Vectorworks icon. The default setting for **Show icon preview** is off (the standard Vectorworks icon displays), and can be changed by the **View > Show View Options** (or the Cmd+J keyboard shortcut) from the **Finder** menu.

When **Show icon preview** is enabled, the default behavior applies only to the current folder, and to the current folder view (with or without details, for example). Click **all folders** in Finder Preferences to see the thumbnails for all folders. Change the setting for each individual View (list view or column view, for example) to see Quick Look thumbnails in all folder views.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Converting Previous Version Files as a Batch
Migrating from Previous Versions

Screen Resolution

Vectorworks software supports high resolution monitors, including Mac Retina® displays. This not only displays crisp, appropriately sized images, but also allows accurate 1:1 scale measurements and line weights to display on screen.

When the Vectorworks application is open across display devices with different resolutions, the following rules apply:

- multiple document windows open on different devices will each display appropriately for the resolution of the screen on which it displays
- documents that span multiple devices display at the resolution appropriate for the screen on which the majority of the document displays
- documents that are mirrored on multiple displays display at the resolution of the lowest-resolution device

On the Mac, to force the software to open in low resolution, for instance to speed up rendering updates while drawing, before opening the Vectorworks application, Ctrl-click on the application icon and select **Get Info**. Select **Open in Low Resolution**. This setting persists across sessions until it is changed.

Closing a File

To close the current file:

1. Select **File > Close** or click the close box on the drawing window.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. If the file changed since it was last saved, click **Save** (Mac) or **Yes** (Windows) to save and close the file. Click **Don't Save** (Mac) or **No** (Windows) to close the file without a save.

To close all open files (Windows only):

1. Select **Window > Close All**.
2. If the files changed since they were last saved, click **Yes** to save and close the files. Click **No** to close the files without a save.

Saving a File

The **Save** command saves the open file, which replaces the earlier version of the file. When you save a file for the first time, specify the file name and location. The **Save** command is disabled if no changes have been made to the file since it was first opened or created.

On Windows, the files require a ".vwx" or ".mcd" extension to be recognized by the Vectorworks program. If you do not enter an extension, the program adds the .vwx extension automatically.

To save the current file:

1. Select **File > Save**.

   If the file has not been saved before, the Save Vectorworks Drawing dialog box opens.
2. Enter a name for the file in the **Name** field, and then select the destination for the file.
3. Click **Save**.

   The program saves the file. The time required may depend on the file size.

Save As

To save the current file with a different name:

1. Select **File > Save As**.

   The Save Vectorworks Drawing dialog box opens.
2. Enter a new **Name** for the file, and then select the destination for the file.
3. Click **Save**.

   The program saves a new copy of the file. The time required may depend on the file size.

Save a Copy As

To save a copy of the current file and continue to edit the current file:

1. Select **File > Save A Copy As**.

   The Save dialog box opens.
2. Enter a new **Name** for the file, and then select the destination for the file.
3. Click **Save**.

   The program saves a new copy of the file and keeps the original file open for further edits. The time required may depend on the file size.

Automatically Saving Files

The autosave feature automatically saves the current file after either a specified number of minutes or a specified number of operations, as defined in the Autosave tab of the Vectorworks Preferences dialog box. See "Autosave Preferences" on page 55 for more information.

If the **Confirm before save** preference is selected, a confirmation dialog box displays before the autosave is performed. The information that displays in this dialog box depends on the settings selected in Vectorworks preferences.

Click to show/hide the parameters.

Parameter	Description
Yes	Saves the file; if the document has not been saved, the Save dialog box opens to name the file first
No	Cancels the save operation and resets the autosave counters
Settings	Opens the Vectorworks Preferences dialog box to edit the autosave parameters; select **Don't backup this document for the remainder of the session** to disable the autosave feature for the active document for the remainder of the session

When the save is complete, the message **Autosave complete** displays in the lower right corner of the window; the message clears when you click the mouse.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Reverting to the Last Saved Version

Reverting to the Last Saved Version

The **Revert to Saved** command replaces the active drawing with the most recently saved version. This deletes all changes made since the last save, which can remove several undesirable changes in one step.

To revert to the last saved version:

1. Select **File > Revert to Saved**.

   This action cannot be undone. Confirm that changes should not be saved.

2. Click **OK**.

   The program closes the active drawing and opens the last saved version of the file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Saving a File

Exiting Vectorworks

To close the Vectorworks program along with any open files:

1. Select **File > Quit** (Windows) or **Vectorworks > Quit (**Mac).
2. If there are any unsaved files, click **Save** (Mac) or **Yes** (Windows) to save the changes and exit. Click **Don't Save** (Mac) or **No** (Windows) to exit without a save.

Converting Previous Version Files

The Vectorworks program must translate files created in previous versions into the current format. The program cannot convert MiniCAD 6 or earlier files, however.

- To convert an entire folder of files at once, use the **Batch Convert** command as described in "Converting Previous Version Files as a Batch" on page 28.
- To convert one file at a time, open and save the file with the current version of the program as described in "Converting a Single Previous Version File" on page 31.

The program automatically applies PNG compression to images contained in converted files.

You may need to correct the visibility of crop objects and annotation objects in sheet layer viewports (see “Creating Sheet Layer Viewports” on page 1616).

For files earlier than Vectorworks 2009, the geometry of the drawing objects is automatically converted using the new Parasolid kernel. The conversion may fail for some solids for a variety of reasons: for example, if an object has self-intersecting profiles inside an extrude or sweep, or if an object’s surfaces do not pass the Parasolid checks. If this happens, the program converts all of the other objects and alerts you that some objects were not converted. A wireframe representation of the failed solid is included in the file; you may be able to edit these objects and regenerate the solids.

As of Vectorworks 2013, dash styles are line type resources. Standard dash styles from previous versions are now included as default resources. At translation time, new resources are created for all custom dash styles that exist in a document, and all objects that use dash styles are updated to use the appropriate resource.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Migrating from Previous Versions
Converting Previous Version Files as a Batch
Converting a Single Previous Version File
Migrating Site Models from Previous Versions
Resetting Plug-in Objects from Previous Versions
Migrating Rendered Files from Previous Versions of Renderworks

Migrating from Previous Versions

The **Update Plug-in Objects** command converts a variety of legacy objects to the current format. This command works across all file layers, and as applicable, on objects that are freestanding, inserted in walls, in a placed symbol, or in a symbol definition. All applicable parameter values, and any attached records are also transferred during the conversion.

Saving a copy of the file prior to running the command is recommended, as the command cannot be undone.

To update plug-in objects:

1. Open a file containing legacy objects.
2. Select **Tools > Utilities > Update Plug-in Objects**.

   The Update Plug-in Objects dialog box opens.
3. Select the type(s) of objects to convert to the new format, and then click **OK**.

   All older version objects of the selected type(s) are converted into the new format.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Converting Previous Version Files

Converting Previous Version Files as a Batch

During a batch conversion, the program scans all files in the source folder (and in subfolders, optionally), and converts previous version files to the current format. Non-Vectorworks files and Vectorworks files that are too old to convert are skipped and recorded in the log file.

There are two options for Vectorworks files during batch conversions:

- By default, the program moves the previous version files to an archive folder and places the converted versions of those files in the source folder.
- Alternatively, the program makes no changes to the source folder. Instead, the converted files (and copies of any current version Vectorworks files) are placed in a specified folder.

To convert a batch of files:

1. Place all files to be converted in one folder, or in subfolders within the source folder.
2. In the Vectorworks program, select **File > Batch Convert**.

   The Batch Convert dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Source Folder	Click **Choose** to open either the Choose Source Folder dialog box (Mac) or the Browse for Folder dialog box (Windows). Select the folder that contains the files you want to convert, and then click **Choose** (Mac) or **OK** (Windows) to return to the Batch Convert dialog box.
Destination Folder	To place the converted files into a different folder, click **Choose** to open either the Choose Destination Folder dialog box (Mac) or the Browse for Folder dialog box (Windows). Select the folder, and then click **Choose** (Mac) or **OK** (Windows) to return to the Batch Convert dialog box. Then deselect **Convert in place**.
Include subfolders	Converts the files within the subfolders of the source folder
Convert in place	Moves the original Vectorworks files to an archive folder, and places the newly converted files in the original source file locations. The archive folder is created within the source folder and is named "Old Version Vectorworks Files." Deselect this option to place the converted files in the destination folder and make no changes to the source folder.
Convert layer links into viewports (Vectorworks Design Series required)	Creates viewports on design layers that reference one or more layers within the same file. Select this option to convert any layer links in the older version files into viewports.
Convert referenced layers into referenced viewports (Vectorworks Design Series required)	Creates viewports on design layers that reference one or more layers in an external file. Select this option to convert any referenced layers in the older version files into viewports.
Convert absolute references into relative references (Vectorworks Design Series required)	Converts workgroup references with absolute file paths to relative file paths, if a relative file path is allowed. A relative file path is not allowed if the file is on a different volume from the referenced file.

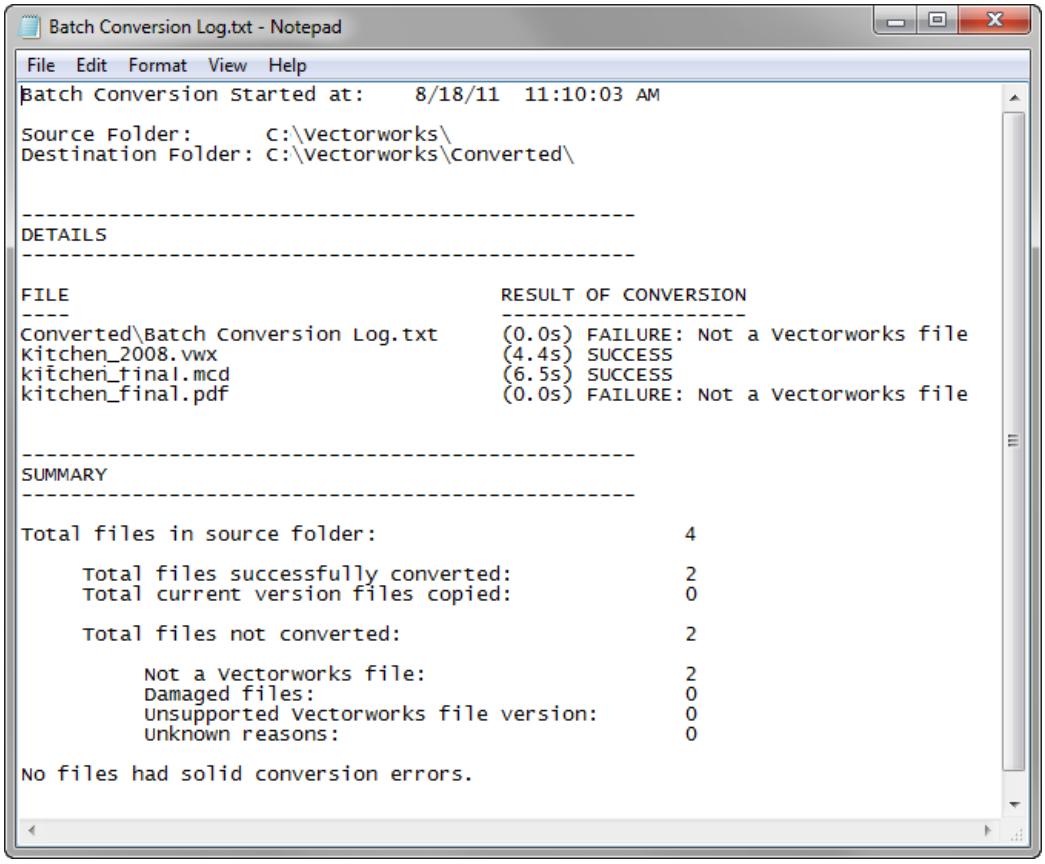
3. Click **OK** to convert the files.

   Each file is briefly opened and re-saved into the appropriate folder. Subfolder names and hierarchies are recreated, if they are included in the conversion process. If **Convert in place** is selected, the original files are moved to the archive folder, in the original subfolder structure.
4. Once the conversion is complete, the Batch Conversion Results dialog box displays a summary.

Click to show/hide the parameters.

Parameter	Description
Total files in source folder	The total number of files contained in the specified source folder
Total files successfully converted	The total number of files successfully converted by the **Batch Convert** command
Total current version files copied	The total number of files that were already in the current version of the Vectorworks program; if **Convert in place** is selected, the files are left in place in the original folder; if **Convert in place** is not selected, the files are copied to the destination folder
Total files not converted	The total number of files that were not converted
Not a Vectorworks file	The number of files that were not converted because they were not Vectorworks files
Damaged files	The number of files that were not converted because they damaged in some way and could not be opened and processed
Unsupported Vectorworks file version	The number of files that were not converted because they were MiniCAD 6 or an earlier version, which is not supported by the **Batch Convert** command
Unknown reasons	The number of files that were not converted, for which the program could not detect a reason
One or more files had solid conversion errors	This message indicates that one or more of the successfully converted files contained a solid that could not be properly generated by the Parasolid kernel. Check the log file to see which files had solid conversion failures. You may be able to edit the converted geometry and regenerate the solids.

5. Click **Details** to view a log file with information about each file processed during the conversion.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Converting a Single Previous Version File
Creating Design Layer Viewports

Converting a Single Previous Version File

When you open an older file, the program automatically converts the file and assigns it an appropriate name. By default, the program renames the converted file and keeps the original file intact. Change the default setting to rename the original file and use the original file name for the converted file instead. In this case, if the original file is write-protected, the program cannot rename it; save the converted file and give it an appropriate name.

To convert a single file:

1. In the current version of the Vectorworks program, open a file that was created with a previous version of the program.

   A conversion alert box opens, showing which file will be renamed, along with the new name. The action described in the dialog box depends on the current setting for converting file names.

2. To change the way this file will be converted, click **Settings**.

   The Vectorworks dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Keep the original file name with the	Select which file will have the original file name: the original file or the new file
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Convert layer links into viewports (Vectorworks Design Series required)	Creates viewports on design layers that reference one or more layers within the same file. Select this option to convert any layer links in the older file into viewports.
Convert referenced layers into referenced viewports (Vectorworks Design Series required)	Creates viewports on design layers that reference one or more layers in an external file. Select this option to convert any referenced layers in the older version file into viewports.
Convert absolute references into relative references (Vectorworks Design Series required)	Converts workgroup references with absolute file paths to relative file paths, if a relative file path is allowed. A relative file path is not allowed if the file is on a different volume from the referenced file.

3. Specify the conversion settings and click **OK** to save the change; future file conversions will use this setting.
4. In the conversion alert box, click **OK** to continue with the conversion.
5. If the file contains a solid that could not be properly generated by the Parasolid kernel, an alert displays to this effect.

   You may be able to edit the converted geometry and regenerate the solid. (See “Converting Previous Version Files” on page 27 for more information.)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Converting Previous Version Files as a Batch
Creating Design Layer Viewports

A L Migrating Site Models from Previous Versions

As of version 12, files containing site model symbols from previous versions can be easily converted.

When you update a site model from a previous version, you are prompted to convert to the new site model. If the file contains more than one site model symbol, an alert dialog box prompts for the selection of one of the symbols to be converted to the primary site model object. The remaining symbols become snapshots of the primary site model.

Site model classes that were necessary for previous versions remain in the file, although they are no longer used.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Sitework Overview

D Resetting Plug-in Objects from Previous Versions

When you migrate plug-in objects from previous versions, some Object Info palette parameters display when they should be hidden. Select the **Reset All Plug-ins** command to update the Object Info palette parameter display.

To reset plug-in objects:

1. Open an older drawing containing plug-in objects.
2. Select **Tools > Utilities > Reset All Plug-ins**.

   The Reset All Plug-ins dialog box opens.
3. Click **Yes** to reset plug-in objects.

   A message displays the update status. When complete, the Object Info palette displays only the pertinent parameters.

Migrating Rendered Files from Previous Versions of Renderworks

Significant changes and improvements to rendering occurred after the Renderworks 2010 version. The Renderworks program now uses the CINEMA 4D® rendering engine, which provides many benefits, such as increased rendering speed, high-quality rendering, additional export options, and a simplified interface.

When opening previous version files, please keep the following points in mind.

- The rendered appearance and quality of converted files is kept as similar as possible. However, some shaders cannot be matched exactly and render differently.
  - The following shaders are not matched: Corrugated, Spanish, and Roman roof tiles.
  - The following Artistic Renderworks options are no longer available: Contour, Mosaic, Stipple, Color Wash, and Ink Print styles.
  - Other Artistic Renderworks styles do not render identically in previous version files; however, the number of Artistic Renderworks options has increased.
- Area lights now emit light in both surface directions, not just in the surface normal direction. Therefore, some masking geometry may be required behind the area light to obtain a similar rendered effect to previous versions.
- Lighting and image brightness from previous versions can now be adjusted with the **Emitter Brightness** parameter in the lighting options (**View > Lighting > Set Lighting Options**). If rendering with Custom Renderworks, adjusting the **Image Exposure** parameter may also help match a previous look.
- If you have created custom plug-in objects in previous versions, the shader record numbers have changed.

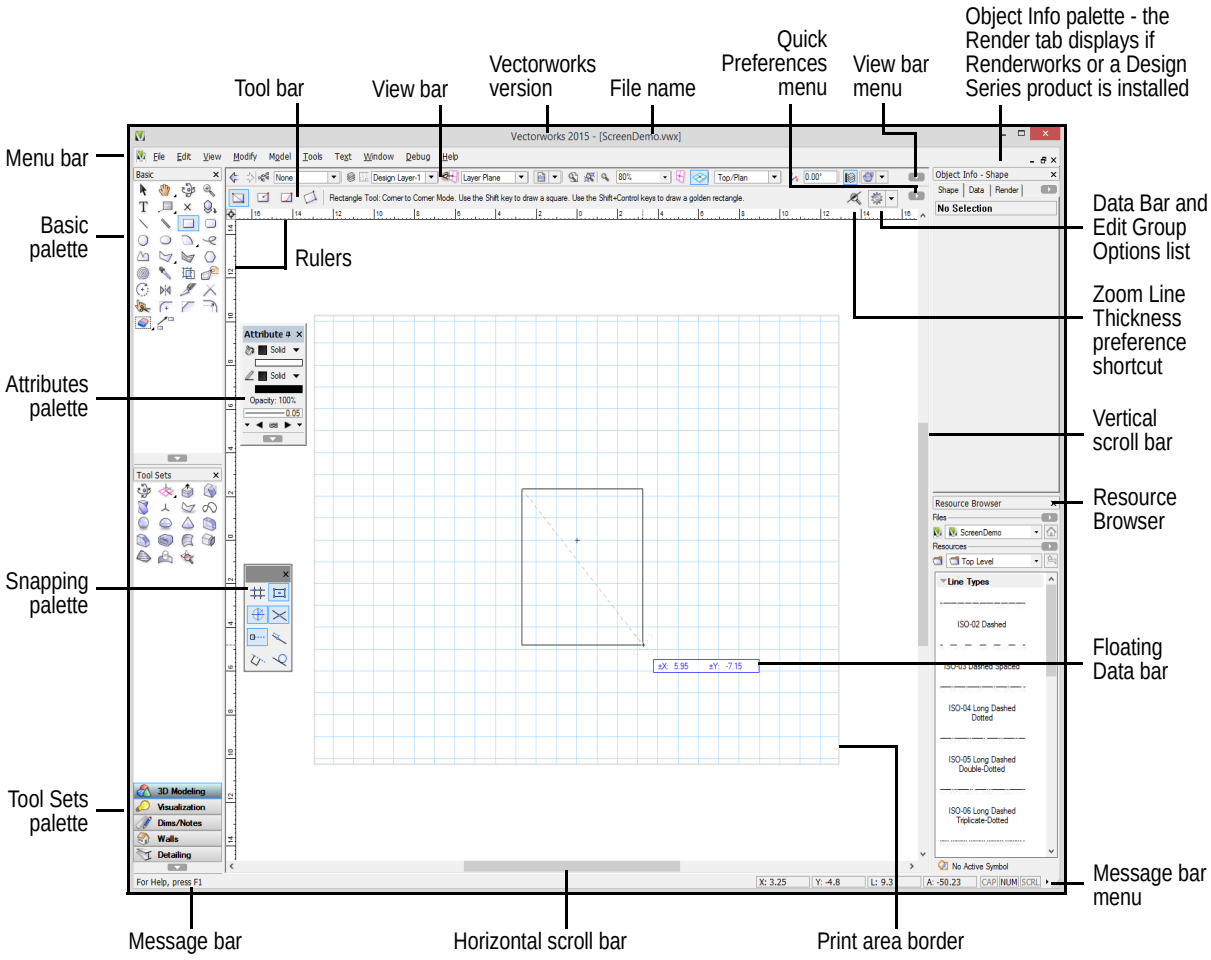
Undocumented Plug-in Objects

Certain older plug-in objects have historically not been documented. As these plug-in objects are updated, they will gradually be incorporated into the help system.

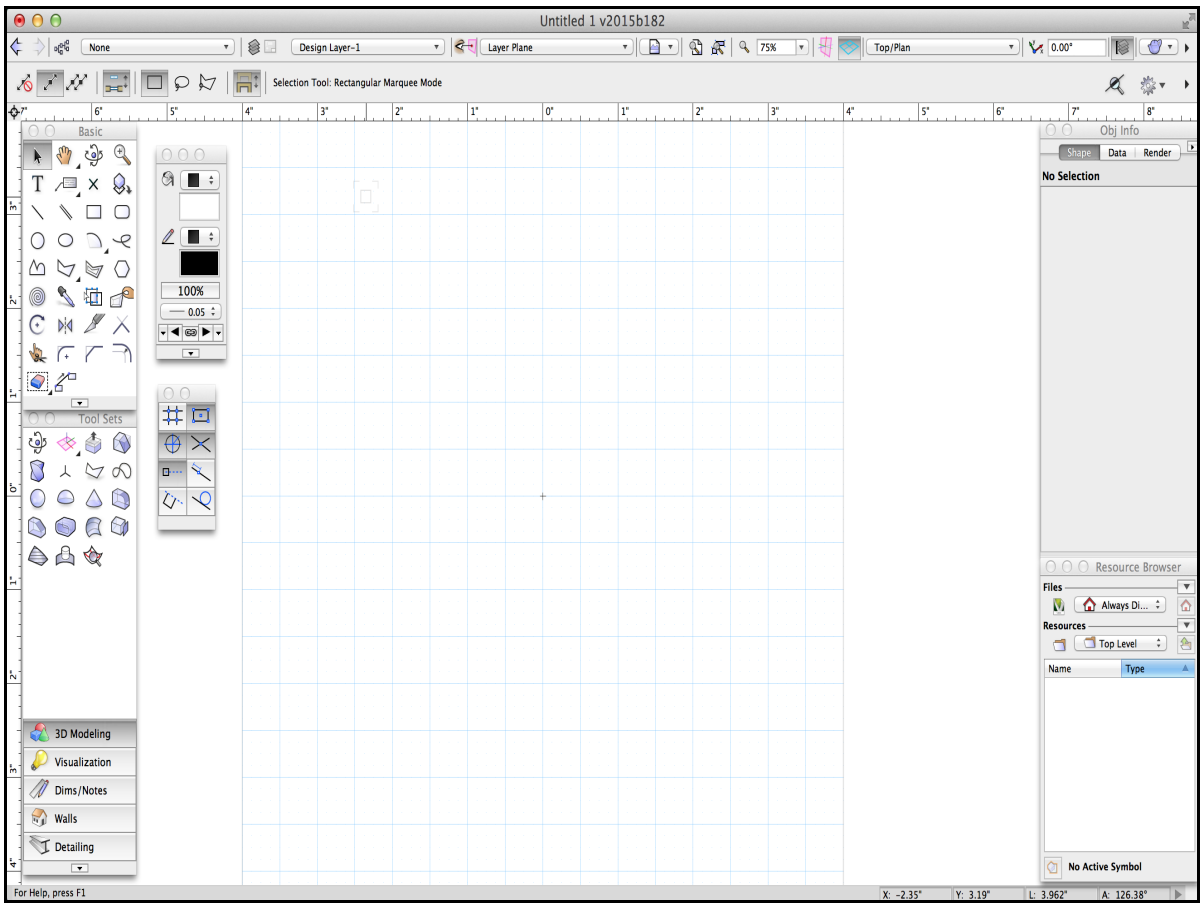
The Vectorworks Workspace

When the Vectorworks Fundamentals program is started, the main window opens with a new, blank file; the Fundamentals workspace is selected, and contains menus, palettes, and tool sets in a default layout. During a work session, palettes may be opened, closed, and moved around as necessary. When the application is closed, the last workspace settings are preserved and restored for the next session.

Custom workspaces can be created, as described in “Creating or Editing Workspaces” on page 1835. To switch to another workspace, select **Tools > Workspaces**, and then select the workspace from the list of those available.



Vectorworks Application Window on Windows



Vectorworks Application Window on Mac

Windows, palettes, tool sets, and dialog boxes that contain a sizing handle in their bottom right corner can be resized; click-drag the sizing handle to the desired location.

The following table describes some of the workspace features common to all Vectorworks products.

Component	Description
Menu bar	Contains pull-down menus that access Vectorworks commands
Title bar	All windows, palettes, tool sets, and dialog boxes have a title bar; click and drag any title bar to move the item to the desired location. See “Palette Layout Options” on page 40.
Data bar	Depending on the tool and on the action being performed, the Data bar displays information such as coordinate data, length, and angle. Use the Data Bar and Edit Group Options list to set the Data bar either to “float” with the cursor, or to be stationary on the Data bar. See “Using the Data Bar” on page 125 for more information. These options can also be set from the selections listed under **Windows > Data Bar Options**.
Tool bar	Displays the various modes of the active tool; click a mode to select it. The bar also displays mode information and accesses the tool preferences, if any. The Tool bar is divided into sections grouped by mode function. To move easily through the mode sections from the keyboard, press the U, I, O, P, [(left bracket), and] (right bracket) keys. Each key corresponds to a consecutive Tool bar section (see the Mode Modifier shortcuts in “Modifying Snapping and Mode Shortcuts” on page 1842).
View bar	Contains buttons and pull-down menus that control the view in various ways (see “The View Bar” on page 37)

Component	Description
Message bar	Displays tool explanations, undo messages, minor alerts, and a progress bar (when applicable). To also display cursor location fields on the Message bar, click the triangle at the far right of the bar; select the option to display all positional fields, only cursor-based location fields, or no cursor location fields.
Drawing area	This is the open portion in the middle of the Vectorworks application window where drawings are created; it includes both the print area and the space that surrounds it
Print area	Within the drawing area, a gray border defines the print area, if shown. Only the objects that are included within the print area are printed. The print area is divided into pages; each page equals a physical sheet of paper to be printed. A print margin is built in for each page (see "The Print Area" on page 1761).
Rulers	Based on the current measurement system, rulers make it easier to precisely create and place objects within the drawing. The 0,0 point on the rulers represents the location of the user origin, or of the working plane axes when using a working plane. The **User Origin** command moves the user origin relative to the internal origin, or moves the working plane origin to that of the layer (see "Setting the User Origin" on page 74). The rulers can be hidden with an option in the Vectorworks preferences (see "Vectorworks Display Preferences" on page 50).
Grids	Based on the current measurement system, two grid systems make it easier to precisely create and place objects within the file. To hide the reference grid, deselect **Show Grid Lines** (see "Snap and Reference Grids" on page 71).

Vectorworks Fundamentals and Design Series Workspaces

When Vectorworks Design Series products are installed, additional workspaces are available in addition to the Fundamentals workspace.

The Fundamentals workspace commands and tools also appear in Design Series workspaces. In some cases, command or tool functionality is extended in the Vectorworks Design Series products, and the location in the workspace may be different. If a command, tool, or parameter described in the documentation requires Design Series, an icon or text indicates that requirement. Examples of commands and tools with extended functionality in Design Series products include straight and round walls, columns, sheet borders, callouts, objects from shapes, and roof object creation.

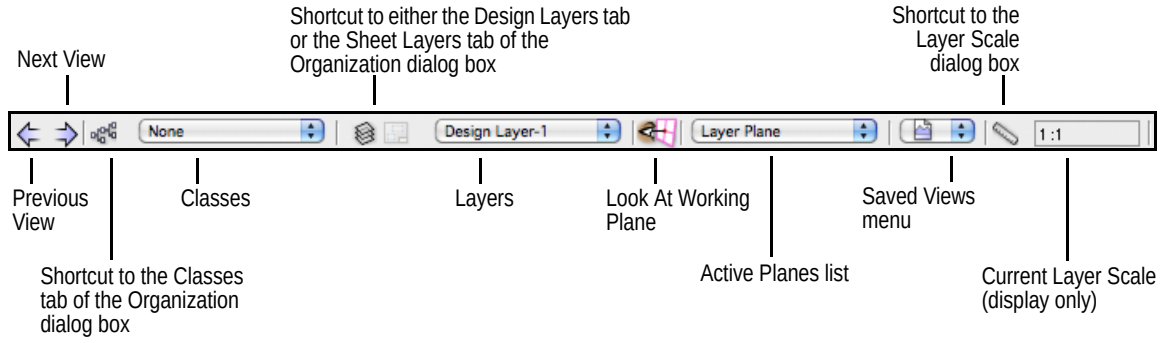
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

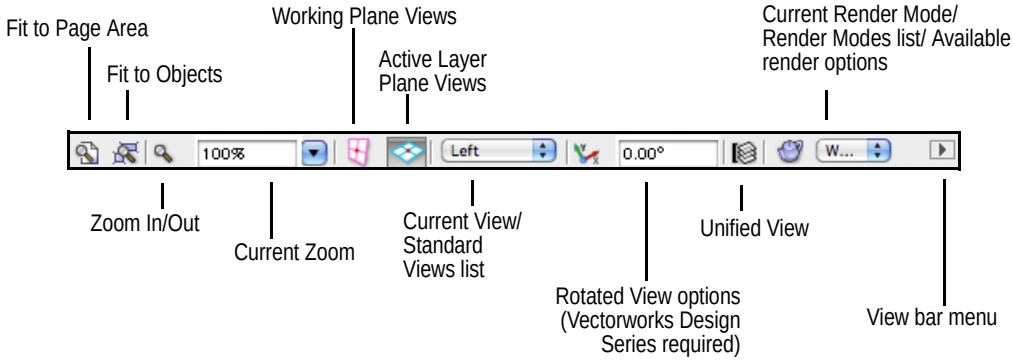
The View Bar
Palettes and Tool Sets
Palette Layout Options
List Box Functionality
Additional Key Functionality
Screen Tips
Object Editing Mode
Setting Vectorworks Preferences
Setting Document Preferences
Context Menus
Using the Resource Browser
The Object Info Palette
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The View Bar

The View bar is located along the top of the application window. It provides quick access to various options that affect the drawing view. Some View bar items have equivalent menu commands.

The selections made in the **View bar** menu determine what displays on the View bar.





Component	Description
Previous View	Displays the previous view that was created by a pan, zoom, or scroll in the drawing area; the application keeps track of up to 50 views
Next View	Displays the next view; the application keeps track of up to 50 views
Classes tab shortcut	Opens the Classes tab of the Organization dialog box
Classes	Displays the active class, activates a class that is selected from the list of existing classes in the file, or enables a new class to be created. When a new class is created, it does not automatically become the active class. The list of classes provides the following information: • An icon to the left of each name indicates the current visibility setting of the class (see "Using Visibility Columns" on page 193 for more information). • A triangle to the right of the class name indicates that the class has subgroups that can be selected individually (for example, a wall class with exterior and interior subgroups). To turn off hierarchical display of classes, see "Session Preferences" on page 52. • On Mac only, if a description was entered for the class, it displays in a screen tip when you position the cursor over the class name.

Component	Description
Design Layers tab/Sheet Layers tab shortcut	Depending on whether a Design Layer or a Sheet Layer is active, opens either the Design Layers tab or the Sheet Layers tab of the Organization dialog box
Layers	Displays the active layer, activates a design layer or sheet layer that is selected from the list of existing layers in the file, or enables a new layer to be created. When a new layer is created, it becomes the active layer. The list of layers provides the following information: • The area to the left of each name indicates the view and visibility of the layer. Depending on the view, any of the following icons can display for design layers; sheet layers are always in Top/Plan view:  Top/Plan view  Any standard view other than Top/Plan  Plan Rotation active (Vectorworks Design Series required) The visibility of the layer is indicated as follows: Black icon — the layer is visible Gray icon — the layer is grayed No icon — the layer is invisible • Referenced design layers display in italics. • On Mac only, if a description was entered for the layer, it displays in a screen tip when you position the cursor over the layer name. If the layer is referenced, the full layer name and the source file name also display in the screen tip.
Active Planes list	Displays the active plane. Depending on the current tool, view, and presence of named working planes, also activates a plane that is selected from the list. See “The Active Planes List” on page 1168.
Look At Working Plane	Changes the view to be perpendicular to the working plane; in other words, rotates the X’, Y’ axes to the screen X and screen Y position
Saved Views menu	Activates a saved view that is selected from the list. Select **Edit View** to open the Saved Views tab of the Organization dialog box, or select **Save View** to open the Save View dialog box; see “Creating or Editing Saved Views Using the Saved Views Menu” on page 192.
Layer Scale shortcut	Opens the Layer Scale dialog box; the scale of the active design layer displays to the right. See “Changing the Scale of the Drawing or the Active Design Layer” on page 67.
Fit to Page Area	Displays the whole drawing (all pages) in the drawing window; see “Fit to Page Area” on page 1162
Fit to Objects	Zooms in or out so that all of the objects in a drawing are visible. If an object or objects are currently selected, the zoom is relative to those object(s); see “Fit to Objects” on page 1162.
Zoom In/Out	Click to double the magnification of the drawing; to reduce the magnification of the drawing by one-half, use Alt-click (Windows) or Option-click (Mac). The zoom centers on any objects that are selected; if nothing is selected, the zoom centers on the last empty spot that was clicked. See “Zooming from the View Bar” on page 105.
Current Zoom	Zooms in or out by the zoom factor that is selected or entered; this option is available when the Zoom - Long option is selected on the **View bar** menu

Component	Description
Working Plane Views	Displays global coordinate views based on X', Y', and Z' axes. In any standard view, and when switching views, such as to Top or Front, the view is based on the working plane position rather than the layer plane position. For a rotated top/plan view (Vectorworks Design Series required), switching to a view such as Front will take into account the rotated plan view and display a rotated Front view.
Active Layer Plane Views	Displays working plane coordinate views (such as Top or Front) based on X, Y, and Z axes. In any standard view, and when switching views, the view is based on the active layer plane position. If switching from a rotated top/plan view (Vectorworks Design Series required), an alert message provides the option to enable Working Plane Views mode instead.
Current View/ Standard Views list	Displays the current view, and activates a standard view (such as Top) that is selected from the list; see "Using Standard Views" on page 1141. If the view is not standard (for example, if the **Flyover** tool was used), Custom View displays.
Rotated View (Vectorworks Design Series required)	Rotates the plan view by the angle specified; see "Rotating the Plan" on page 1163
Unified View	Toggles Unified View mode; see "Unified Layer View" on page 1152
Current Render Mode Options	If the current render mode has options, displays the appropriate options dialog box; this option is available when the Render Mode - Long option is selected on the **View bar** menu
Current Render Mode/ Render Modes list	Displays the current render mode, and activates a render mode selected from the list; select **Options for Other Render Modes** to access the options for a particular mode. See "Rendering with Vectorworks" on page 1587 and "Renderworks Rendering Modes" on page 1593.
View bar menu	Selects the options to display on the View bar

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Layers
Creating Classes
Setting Design Layer Properties
Setting Class Properties
Saved Views
Zooming
Using Standard Views
Rendering with Vectorworks
Renderworks Rendering Modes
Understanding the Working Plane
Rotating the Plan

Palettes and Tool Sets

The Fundamentals workspace contains various palettes for creating and editing objects. Basic and Tool Sets are tool palettes, which have special functionality, as described in "Tool Palette Features" on page 42. If a keyboard shortcut is currently assigned to a menu command or tool, the shortcut displays when the mouse hovers over the command or tool; to set up or modify keyboard shortcuts, see "Modifying Menus and Commands" on page 1837.

Depending on the initial settings, some palettes may be hidden when the program is launched.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Palette / Tool Set	Purpose
Snapping	Contains SmartCursor snapping controls that can be toggled on or off; see “Setting Snapping Parameters” on page 131
Attributes	Contains a selection of colors, fills, pens, and other object attributes; see “The Attributes Palette” on page 1093
Object Info	Lists context-sensitive object information for viewing and editing; see “The Object Info Palette” on page 997
Working Planes	Contains controls for adding and modifying working planes; see “The Working Planes Palette” on page 1176
Resource Browser	Accesses the resources available for use in drawings, including gradient fills, hatch patterns, image fills, record formats, scripts and script palettes, symbols and symbol folders, worksheets, textures, backgrounds, and more. See “Resource Libraries” on page 219.
Visualization (Renderworks required)	Accesses all lights and cameras in the file; see “Managing Lights and Cameras with the Visualization Palette” on page 1611
Basic	Contains a single set of basic object creation and editing tools; the palette can be customized through the Workspace Editor
Tool Sets	In the Fundamentals workspace, the palette includes the following tool sets, whose tools are grouped by similar functionality; the palette and its tool sets can be customized through the Workspace Editor
3D Modeling	Contains tools for creating and editing 3D objects, including solids and NURBS
Visualization	Contains tools for changing the drawing view in different ways, including the **Walkthrough** and **Light** tools
Dims / Notes	Contains tools for adding dimension and label objects
Walls	Contains basic wall creation tools
Detailing	Contains tools for adding architectural detail objects, such as tubing
Script Palettes	Contains palettes with script resources

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Vectorworks Fundamentals and Design Series Workspaces

Palette Layout Options

Customize the palette layout as needed.

- Select **Window > Palettes** to display or hide palettes. Alternatively, right-click (Windows) or Ctrl-click (Mac) on the drawing area, and select **Palettes** from the document context menu.
- Click on the title bar of any displayed palette and drag it to the desired location.
- Most palettes can be resized by the standard Windows or Mac resize method.

Each time you exit the Vectorworks program, the current palette settings and positions are automatically saved for each workspace in a file in your user folder ([User]\Settings\SavedSettings.xml). The settings in this user file override the initial palette settings in the workspace file. To reset to the original workspace settings, click **Reset Saved Settings** from the Session tab of Vectorworks preferences (see “Session Preferences” on page 52).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Specifying Mac Palette Margins

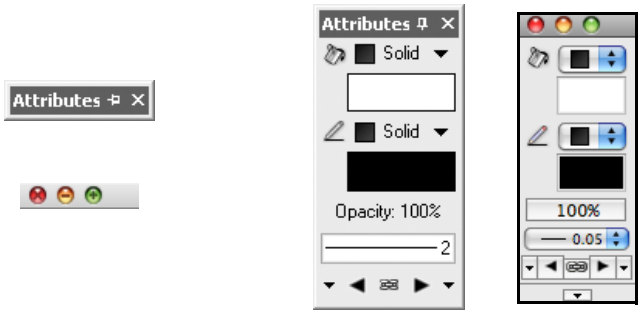
On the Mac, the palettes can be placed in a margin on either side of the drawing area. When the palette margins are in use, the drawing area cannot expand past a fixed size, which allows the palettes to remain out of the way. The palette margin area can be turned on and off in the Vectorworks preferences as described in “Session Preferences” on page 52.

Minimizing Palettes

To increase the available drawing area, palettes can be minimized while not in use, and then maximized when needed.

On Mac, click the palette’s yellow title bar button or double-click the palette’s title bar to minimize the palette. Repeat the process to maximize the palette.

On Windows, click the palette’s pin icon in the title bar to toggle between minimized (horizontal pin icon) and maximized (vertical pin icon) display. Move the cursor over a minimized title bar to temporarily maximize the palette; move the cursor off the palette to minimize it again.



Minimized palette Maximized palette

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Docking Windows Palettes
Tool Palette Features
Vectorworks Fundamentals and Design Series Workspaces

Docking Windows Palettes

In Windows, most palettes can be docked to any of the four drawing window edges, or snapped to other docked palettes. The Object Info, Resource Browser, Attributes palette, and Working Planes palettes can only be docked to the left and right window edges.

The ability to dock palettes is enabled by default in the Session tab of the Vectorworks Preferences dialog box. Deselect this option to disable docking. See “Setting Vectorworks Preferences” on page 49 for more information.

To dock or undock a palette by double-clicking:

1. Double-click a palette’s title bar.

   If the palette was undocked, double-clicking docks it in its previous location. If the palette was docked, double-clicking undocks it and moves it to its previous location in the drawing area.

2. To toggle the docked/undocked status, double-click the palette’s title bar again, as necessary.

To dock a palette by dragging it:

1. Drag the title bar of the desired palette toward the edge of the window.

   The palette’s view switches to a gray outline. When the palette is in range of a window edge, the outline changes shape to represent the new docked shape.

2. Move the palette’s outline to the edge of the docking location and release. Currently docked palettes adjust their location along the edge to accommodate the new palette.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To prevent a palette from docking, hold down the Ctrl key while dragging the palette near an edge.

To undock a palette by dragging it:

1. Drag the title bar of the desired palette away from the edge.

   The palette's view switches to a gray outline. When the palette is out of window edge range, the outline changes shape to represent the new undocked shape.

2. Drag the palette to the desired location within the drawing area.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

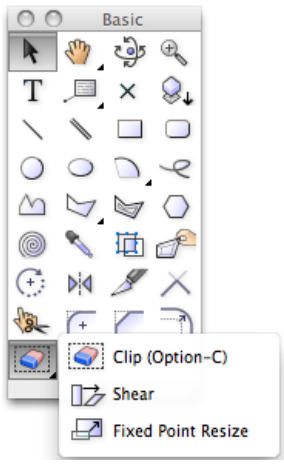
Palette Layout Options
Tool Palette Features
Vectorworks Fundamentals and Design Series Workspaces

Tool Palette Features

Both the standard tool palettes (Basic and Tool Sets) and any custom tool palettes have special features that other palettes do not. Tool palettes and their tool sets can be created and customized using the Workspace Editor (see "Creating or Editing Workspaces" on page 1835).

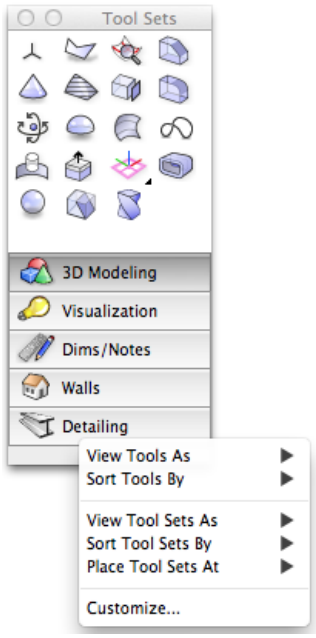
Pop-out Tools

On tool palettes, an arrow on the right side of a tool icon or label indicates additional, related pop-out tools. Click and hold down the mouse button to open the menu of pop-out tools.



Utility Menus

Each tool palette has a button at the bottom that opens a utility menu, which controls the palette and tool display.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

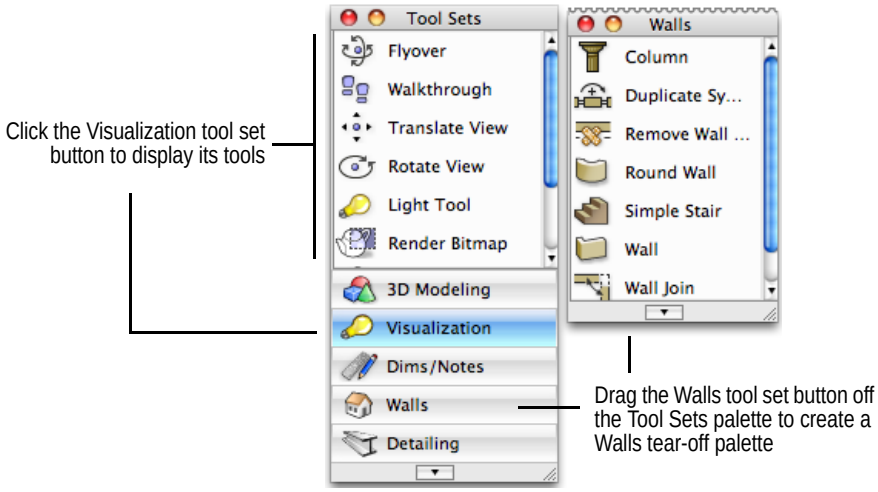


Menu / Command	Action
View Tools As	
Icons	Display only an icon for each tool
Icons and Text	Display both an icon and a text label for each tool
Text	Display only a text label for each tool
Sort Tools By	
Manual Placement	Display tools in the order appearing in the Workspace Editor
Ascending Alphabetical	Display tools in ascending alphabetical order according to text labels
Descending Alphabetical	Display tools in descending alphabetical order according to text labels
View Tool Sets As	(These options are available only for tool palettes with multiple tool sets)
Icons	Display only an icon for each tool set in the palette
Icons and Text	Display both an icon and a text label for each tool set in the palette
Text	Display only a text label for each tool set in the palette
Sort Tool Sets By	(These options are available only for tool palettes with multiple tool sets)
Manual Placement	Display tool sets in the order appearing in the Workspace Editor
Ascending Alphabetical	Display tool sets in ascending alphabetical order according to text labels
Descending Alphabetical	Display tool sets in descending alphabetical order according to text labels
Place Tool Sets At	(These options are available only for tool palettes with multiple tool sets)
Top of Palette	Display tool set selection buttons at the top of the tool palette
Bottom of Palette	Display tool set selection buttons at the bottom of the tool palette
Customize	Opens the Workspace Editor Options dialog box (see “Creating or Editing Workspaces” on page 1835 for details)

Tool Set Selection Buttons and Tear-off Palettes

If a tool palette has multiple tool sets (as does the Tool Sets palette), selection buttons for each tool set display on the palette. To display a tool set's tools, click the appropriate button. The button is highlighted to indicate which tool set is active.

To see multiple tool sets at once, drag a tool set's selection button off the main palette. This creates a separate, temporary palette for the tool set, with a "perforated" top edge. Like other palettes, these tear-off palettes can be docked, resized, and minimized. To close a tear-off palette, click its close button. On Windows, a docked tear-off palette does not have a perforated edge; instead, it has a special close button in the shape of a curved arrow.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

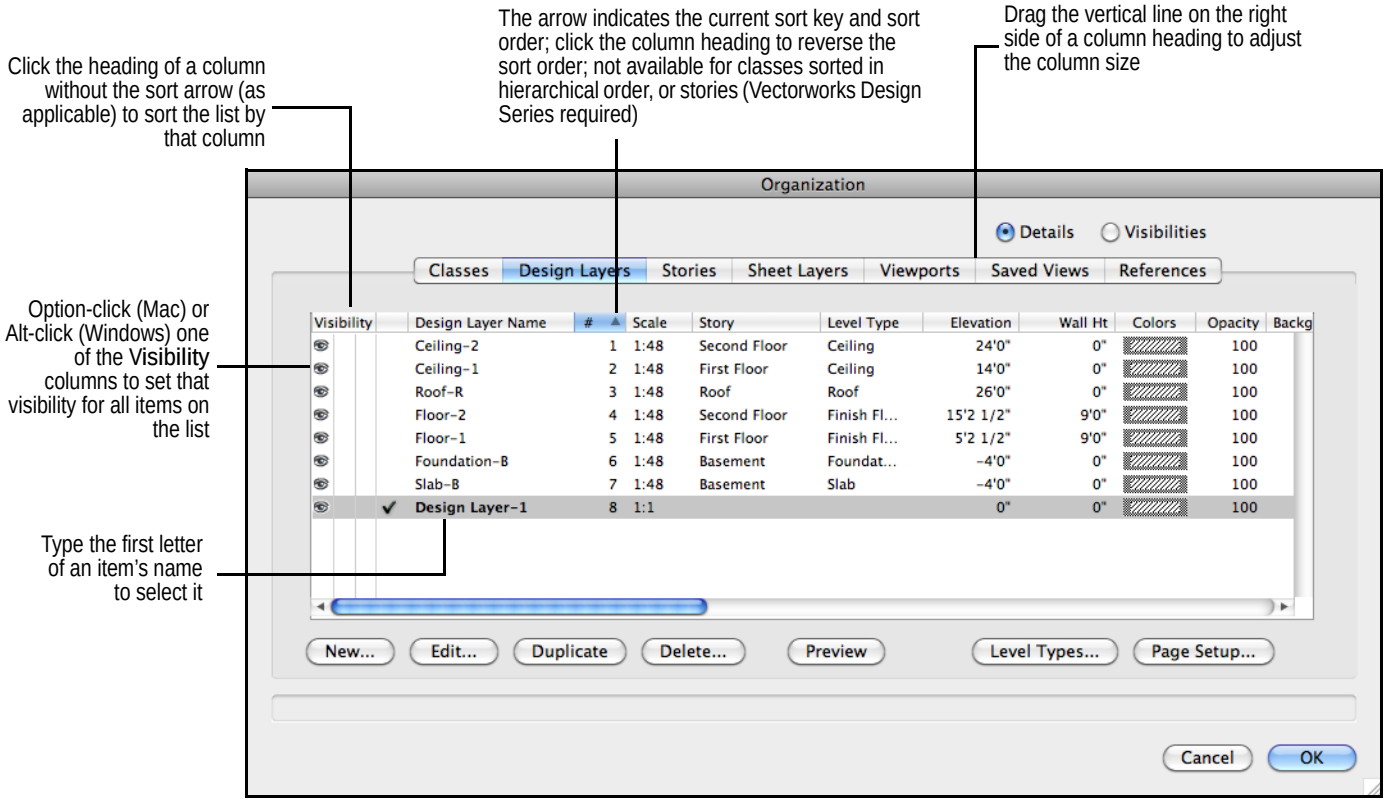
Palette Layout Options
Docking Windows Palettes
Vectorworks Fundamentals and Design Series Workspaces

List Box Functionality

Dialog boxes that contain long lists of information (such as the Organization dialog box) may have some or all of the following functionality.

Task	Action
Change the list's sort key	Click the heading of the column to sort by; an arrow appears on the right side of the column heading to indicate that it is the sort key
Change a column's sort order	Click the column heading; the sort arrow in the heading indicates whether the current sort is ascending or descending
Resize a column	Click the vertical line on the right side of a column's heading and drag it left or right
Select a group of items	Click the first item, and then Shift-click the last item in the group
Select multiple items individually	Click the first item, and then Command-click (Mac) or Ctrl-click (Windows) each additional item
Select an item on the list	Type the first letter(s) of the desired item's name
Edit an item on the list	Double-click the desired item
Open a context menu for an item on the list	Ctrl-click (Mac) or right-click (Windows) the desired item
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Task	Action
For lists with a **Visibility** column, set the same visibility for all items on the list	Option-click (Mac) or Alt-click (Windows) the desired setting
Display lists in hierarchical order	Classes can be displayed in hierarchical order (up to four levels). To enable or disable hierarchical display of classes on pop-up menus including in the Object Info palette, the View bar, and dialog boxes, see “Session Preferences” on page 52. On the Classes tab of the Organization dialog box and the Navigation palette (Vectorworks Design Series required), display options are saved separately. See “Displaying Classes in Hierarchical Order” on page 159.



Vectorworks Fundamentals and Design Series Workspaces
Displaying Classes in Hierarchical Order

Additional Key Functionality

In the Vectorworks program, several keys have additional functionality beyond the standard conventions:

Key	Usage
Esc	Cancels the current operation. When a dialog box is open, this is the equivalent of pressing the **Cancel** button. In the Data bar or Object Info palette, cancels the entry in a field and returns the focus to the drawing area. Cancels rendering. Clears smart points. Deselects the working plane.

Key	Usage
Return (Mac) / Enter (Windows)	When a dialog box is open, this is the equivalent to pressing the **Done** or **OK** button. In a field in the Data bar or Object Info palette, this accepts the information and returns the focus to the drawing area. Adds a line of text to a text block. In the Object Info palette, press Shift+return (Mac) or Shift+Enter (Windows) to save the entry and keep the focus in the same field, so that you can enter a different value if necessary.
Tab	Moves the cursor from one field to the next in the Data bar, Object Info palette, and dialog boxes. Press Shift+Tab to move backwards through the fields.
Delete (Mac) / Backspace (Windows)	Removes the last segment of or cancels the creation of an object being drawn
Ctrl (Windows)	Prevents palettes from docking when they are moved near an edge
Space Bar	Engages the Pause/Boomerang mode. Holding down the Space Bar temporarily pauses the current tool. A second tool can then be selected and used. Release the Space Bar to return to the previous tool.
Command (Mac) / Shift (Windows)	Briefly expands the currently selected screen tip

Using Arithmetic Expressions

Arithmetic expressions can be entered into most of the edit fields, including the Object Info palette's Shape tab and the Data bar. For this reason, the dash (-) cannot be used as a separator between feet and inches.

Parentheses can be used to override the default operator precedence. For example:

1" + 2" * 3" = 7" (without parentheses)

(1" + 2") * 3" = 9" (with parentheses)

Values can be entered in any unit; the program converts it into the current unit. For example, if the current unit is Inches and an entry of 4"+3 cm is made into the Object Info palette X field, the program converts the units automatically. The result, 5.1811", is displayed.

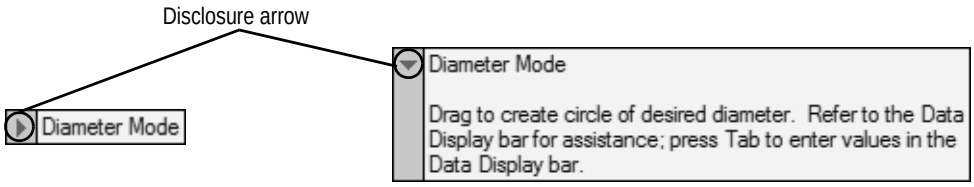
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Units
Vectorworks Fundamentals and Design Series Workspaces

Screen Tips

Screen tips are available throughout the program to identify items such as tool, mode, and snapping names. To view a screen tip, hold the cursor briefly over the item in question.

On Windows, some items have screen tips that can be collapsed (default) and expanded. When collapsed, only the name of the item displays. When expanded, additional help information is displayed below the name. To expand screen tips, click on the disclosure arrow. When a screen tip is expanded, it remains expanded until it is manually collapsed again. Screen tips for all other items are expanded until a tip is collapsed again.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Collapsed screen tip

Expanded screen tip

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Vectorworks Fundamentals and Design Series Workspaces

Context Menus

Click on an object, plug-in object, the drawing area, or a section of the Resource Browser with a right-click (Windows) or Ctrl-click (Mac), to display a menu containing context-sensitive commands that pertain to the selected item or items. If several similar items are selected, only the applicable context menu commands to all the items in the selection become available. Many of these commands can be used to move directly from one editing mode to another, bypassing a dialog box. For instance, context menu commands can be used within an edit viewport mode to switch directly from editing one viewport component to another, such as from editing a crop to editing annotations.

Document and object context menus can be customized to include frequently-needed commands; see “Modifying Context Menus” on page 1839.

Many context menu commands are also available from the menu command lists; however, certain commands are available exclusively from a context menu.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Preferences

Setting Vectorworks Preferences

Vectorworks preferences are options that apply to every file you open, every time you run the program.

To change the Vectorworks preference settings:

1. Select **Tools > Options > Vectorworks Preferences**.

   The Vectorworks Preferences dialog box opens. There are seven preferences tabs (Edit, Display, Session, 3D, Autosave, Interactive, and User Folders).

2. Click one of the tabs to set the Vectorworks preferences for that tab.
3. Click **OK** to save the settings.

Edit Preferences

Click the Edit tab to set preferences that control various edit functions in the program.

Click to show/hide the parameters.

Parameter	Description
Click drag drawing	Lets you draw objects with the click-drag method rather than the click-click method (click-click is the default). Click once with the mouse button and do not release; drag the cursor to a desired location and then release. See “Using the Mouse” on page 103.
Eight reshape handles	Shows eight handles (four corner and four center) on most objects; deselect the option to display only four corner handles
Offset duplications	When you use the **Duplicate** command, places the duplicate object so that it is offset from the original object; deselect the option to place duplicates directly over the original
Auto join walls	When you use the **Wall** tool, automatically joins walls at corners and intersections; when walls are separated, their ends automatically heal; when walls have core components, components also automatically join (see “Automatically Joining Walls” on page 542)
Separate sheet views	Saves the view origin and zoom factor for each sheet layer; deselect the option to use the same view for all design layers and sheet layers
Mouse wheel zooms	Sets the default behavior of the mouse wheel and the scroll ball. When selected, the wheel zooms by default; when deselected, the wheel scrolls by default. See “Zooming with the Mouse Wheel” on page 105 and “Scrolling with the Mouse Wheel” on page 107 for details.
Allow ctrl-click/ option-click in-place duplication	When selected, allows a Ctrl-click (Windows) or Option-click (Mac) with the **Selection** tool to create a duplicate copy of a selected object in place. Since it is simple to inadvertently create duplicate objects that cannot be easily detected, leaving this option deselected is recommended; it is deselected by default. Regardless of the setting made here, duplicates can be created with a Ctrl-click/drag or Option-click/drag.
Edit text horizontally by default	Always displays the text editing box in a horizontal position, even when the text is rotated (see “Creating Rotated Text” on page 385)
2D conversion resolution	Sets the number of segments that will be used to represent polylines and circles when you draw and edit objects
Default compression	Specifies the default image compression to apply to images in a viewport cache and to images created by the **Render Bitmap** tool (Renderworks required). PNG compression provides the best image quality but produces larger files, while JPEG creates smaller files, but with possible loss of detail. PNG is selected by default.

Parameter	Description
Allow interactive 2D symbol scaling	Specifies whether the **Selection** tool, in Interactive Scaling mode, is allowed to resize symbol instances; if allowed, a warning can be displayed to avoid inadvertently scaling symbol. If Never is selected, symbols can still be resized from the Object Info palette; see “Scaling Symbols from the Object Info Palette” on page 1064.
Arrow Keys	Select an arrow and modifier key combination as a shortcut for four common operations; each key combination can be assigned to only one operation. Alternatively, disable the ability to use arrow key combinations to perform any of these functions.
Switch active layer/class	Select the key combination that switches the active layer (up and down arrows) and the active class (left and right arrows)
Pan drawing	Select the key combination that pans the drawing by half of the area currently in view
Nudge objects	Select the key combination that nudges objects by one pixel, and select whether to display an alert on the Message bar when nudging items to avoid accidentally moving objects
Move objects	Select the key combination that moves objects, and select whether to move objects by the snap grid distance or by the specified custom distance

Vectorworks Display Preferences

Click the Display tab to set the display preferences.

Click to show/hide the parameters.

Parameter	Description
Rulers	Shows the rulers
Show Internal Origin Marker 	Displays the drawing’s internal origin in Top/Plan view. The marker does not print.
Colored axes in Top/Plan view	Displays X (red) and Y (green) axes in 2D Top/Plan view
Scroll bars	Shows the scroll bars
Black background	Uses a black drawing background instead of the default settings
Zoom line thickness	Lines in the drawing appear thicker when you zoom in; the screens redraw slower on Mac or on Windows if GDI+ imaging is enabled
Create text without fill	When you create text objects, dimension objects, and some plug-in objects that contain text, the text block has no fill, even if the default attributes are set to add a fill to new objects. This prevents objects beneath the text from being obscured.
Show parametric constraints	Displays parametric constraints

Parameter	Description
Show other objects while in editing modes	When certain complex objects are edited, an Edit window with a colored border displays in the drawing area. When you edit solids, groups, symbols (in the drawing only), or viewports in the Edit window, select this option to make the other objects from the drawing visible and snappable during editing; deselect the option to show only the object being edited. (See “Object Editing Mode” on page 1004 for more information.) This option does not apply to other types of objects that are edited from the Edit window, including extrudes, multiple extrudes, tapered extrudes, sweeps, meshes, floors, and roof faces. For symbols, the option applies only when you edit a symbol definition by clicking on a symbol instance in a design layer.
Gray other objects	If **Show other objects while in editing modes** is selected, select this option to gray the objects in the drawing that are not being edited
Use VectorCaching for faster drawing	Caches vector information for complex document entities, such as polylines and hatches; while this makes screen redraws faster, it also can potentially increase RAM requirements by up to 50 percent
Center on objects after view change	When switching standard views (such as Top or Left Isometric), centers the view to display either selected objects, or if nothing is selected, all objects on the layer or in the drawing. Centers the view on all objects in a layer if there are no objects in view after switching between layers. This helps you feel oriented when switching views or changing layers.
GDI+ imaging (Windows only)	Draws lines of equal thickness with round end caps; also provides these features: • fills in 3D planar objects • variable object opacity • layer transparency • anti-aliasing • object transparency in X-ray Select mode Disable this option if you need to use OpenType fonts.
Anti-aliasing	Blends the edges of fills and lines for a smoother appearance; on Windows, this option is available only when GDI+ imaging is enabled
Use Enhanced Navigation Graphics	This preference provides more graphical information as you navigate in your drawing (for example, during pan, zoom, flyover, and walkthrough). It also provides more on-screen clarity and detail as you navigate between views. If your hardware cannot support this feature, it can be deselected.
Display light objects	Controls the visibility of light objects; hide light objects to reduce screen clutter but maintain light effects
Always	Light objects are always visible
Only in wireframe	Light objects are only visible in Wireframe mode; otherwise, they are hidden
Never	Light objects are hidden regardless of the render mode
Display 3D loci	Controls the visibility of 3D loci
Always	3D loci are always visible
Only in wireframe	3D loci are only visible in Wireframe mode; otherwise, they are hidden
Never	3D loci are hidden regardless of the render mode

Parameter	Description
Edit Font Mappings	Specifies the replacement fonts for fonts that are not available. Font mappings can be changed or deleted. If mappings are deleted, the Font Mappings dialog box appears so that unavailable fonts can be mapped.

Session Preferences

Click the Session tab to set general Vectorworks preferences.

Click to show/hide the parameters.

Parameter	Description
Use sound	Supplements the visual SmartCursor cues with audible cues; move the slider bar left (to reduce) or right (to increase) the program volume relative to the system volume
Log time in program	Records in a log file the time spent in the program, as well as the time spent to open and close documents; the date format depends on the language and regional settings in the operating system. The log file is called Vectorworks Log.txt, and it is created in the User Data and Preferences Folder (which displays on the User Folders tab of Vectorworks preferences).
Display minor alerts on message bar	Displays minor warnings on the Message bar instead of in a dialog box
Display classes in pop-up menus hierarchically	Displays classes hierarchically in pop-up menus including in the Object Info palette, the View bar, and dialog boxes throughout Vectorworks. Hierarchical class display in the Organization dialog box and Navigation palette (Vectorworks Design Series required) is controlled separately, on the Classes tab of the dialog box/palette.
Run scripts in developer mode	For plug-in development, controls the display of warning messages during script execution. This is a convenient way for advanced developers to speed up the development process. By default, the scripts of scripted plug-in objects, tools, and menus are compiled once and stored in memory. The object, tool, or menu command memory cache is executed when the script is invoked. However, in developer mode, the script is compiled each time so that script changes are executed immediately and the developer does not need to restart the application to view changes. For Python scripts, warning messages are displayed along with Python log data, and initialization code (extended properties set in the script options) are called on each run. For VectorScripts, plug-ins are compiled on each run and warning messages are displayed. Plug-ins may work more slowly when this option is enabled.
Maximum number of undos	Sets how many undo operations are kept in memory; the maximum number of undos is 100
Issue undo warnings	Presents a dialog box when an action that cannot be undone is about to be performed

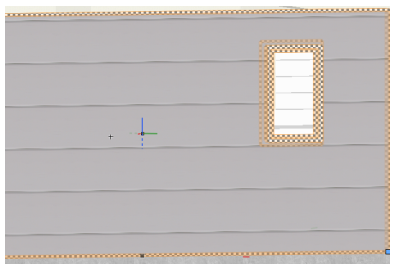
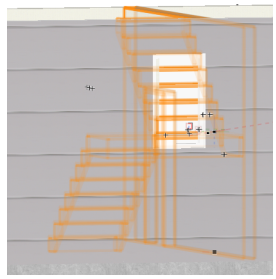
Parameter	Description
Undo view changes	Sets how the program handles view changes when you undo actions • Never - ignores all operations that are strictly view changes • Grouping All View Changes - treats all consecutive view changes as one single undoable action • Grouping Similar View Changes - treats similar consecutive view changes as a single undoable action • Individually - treats each individual view change as an undoable action
Enable palette docking (Windows)	Lets you dock palettes; deselect the option to disable docks and to undock all active palettes
Automatically append file extensions (Mac)	Appends the appropriate file extension (.vwx, or .sta for template files) to a newly created drawing file on a Mac
Display default content	Enables pre-defined content (such as hatches and gradients) to display for selection throughout the program
Create a new document on startup	Creates a new, untitled file when the program is launched. The new file is based on the Default.sta template file (if one exists); otherwise, it is blank.
Change active layer for Similar Object Creation (Vectorworks Design Series required)	When the **Create Similar Object** command is invoked, changes the active layer to be that of the source object. Select this option to create the new object on the same layer as the source object. See “Creating Similar Objects” on page 124.
Use local help even when an Internet connection is available	When you are connected to the Internet, the help system is obtained online. When you are not connected, the local version of the help displays. There is not normally a big difference between the online and local versions. If bandwidth or connection issues cause online help display or searching to become unacceptably slow, select this option to always use the local version of the help system.
Enable Mac OS restore windows functionality for Vectorworks (Mac)	For Mac OS X 10.7 and later, enables the operating system to automatically restore previously open files when re-launching Vectorworks. Keep this option deselected when working with several different versions of Vectorworks, to prevent older versions from unsuccessfully attempting to open newer version files. This option is not available on Windows or versions of the Mac operating system prior to 10.7.

Parameter	Description
Check for updates	Select how often the program should check for available software updates. When the specified update check interval has elapsed, an alert dialog box displays if a new Service Pack (maintenance release) or major release of the Vectorworks software is available. If both a Service Pack and major release are available simultaneously, the system will send notification about the Service Pack first, followed by notification about the major release in the next session. • Click **More Info** (or **Tell Me More**) in the alert dialog box to open a web browser to display and download the available update. You must exit the program to successfully install the software update. • If it is not convenient to update the software at this time, click **Not Now** (or **Remind Me Later**) in the alert dialog box. The alert dialog box will redisplay when the specified update check interval has elapsed. • To disable notifications of a major release until the next major release is available, click **Not Interested**. If the program is unable to connect to the update server, it attempts to connect once a day for seven days after the original failed attempt. If a connection to the update server still cannot be established, a dialog box opens to suggest running a manual update. If the **Check for Updates** command is run after the automatic update check fails, the next automatic update check occurs the day after the manual check.
Error reporting	Optionally sends information about application crashes and other basic usage statistics to Nemetschek Vectorworks. Crash details allow our programmers to receive crash notifications, including the area of the program where the crash occurred. Usage patterns help us see what events led up to a particular crash.This reporting assists us with product development while maintaining user privacy; to allay any concerns you may have, we never see the data in your drawings, and the reporting has no impact on your software operations.
Serial Numbers	Opens the Serial Numbers dialog box to add or remove serial numbers for all installed Nemetschek Vectorworks products (see “Adding and Removing Serial Numbers” on page 21)
Reset Saved Settings	Reverts to default settings instead of user-specified settings for tool modes, dialog box positions, and dialog box values. For palette positions and settings, reverts to the settings established when the workspace was created (in the User Data and Preferences folder). In the dialog box that opens, select whether to reset settings for always performing the selected action in alert dialog boxes. Also select whether to reset settings for all tool modes, dialog box positions, dialog box values, and palette positions and settings. Click **OK** to return to the Vectorworks Preferences dialog box.
Palette margins (Mac)	Sets whether the document window leaves a space for palettes when the window is opened

3D Preferences

Click the 3D tab to set preferences for 3D edits.

Click to show/hide the parameters.

Parameter	Description
3D Rotation	Sets the detail level that displays while the 3D view is rotated. Select Detailed to display objects completely, but to rotate slower. Select Responsive to rotate faster, but to display objects with less detail during the rotation.
Retain Rendering Model	Determines the degree to which the program retains the rendered model in memory during 3D rotation. Select Never to force the model to always display in Wireframe mode; select Always to force the model to remain rendered.
3D conversion resolution	Sets the segmentation resolution used to display curved 3D objects; affects extruded and swept polylines, circles, and arcs
Render mode when changing from Top/Plan to a 3D View	Sets the preferred 3D render mode to OpenGL or Wireframe; this setting is applied whenever a drawing is switched from Top/Plan to a 3D view
Projection when changing from Top/Plan to a 3D View	Sets the preferred 3D projection to Orthogonal, Narrow Perspective, Normal Perspective, or Wide Perspective; this setting is applied whenever a drawing is switched from Top/Plan to a 3D view
Use occluded selection and snapping	In rendered views, solid objects "occlude," or hide, objects behind them. Keep this option enabled to more easily select, and snap to, solid objects without interference from hidden geometry. To access hidden geometry, use the X-ray Select mode of the **Selection** tool, turn on the clip cube, or switch to Wireframe render mode; see "X-ray Select Mode" on page 113.  Occluded selection and snapping enabled  Occluded selection and snapping disabled

Click here for a video tip about this topic (Internet access required).

Autosave Preferences

Click the Autosave tab to set preferences for automatic file saves and backups.

Click to show/hide the parameters.

Parameter	Description
Autosave every	Enables the Autosave feature; also sets the number of minutes or operations between autosaves
Confirm before save	Before each save, opens a dialog box so that you can choose either to save or to continue to work without a save. The timer/counter resets regardless of which option is selected. See "Automatically Saving Files" on page 26.

Parameter	Description
Overwrite original file	Writes over the original file with the latest changes during a save
Autosave a backup copy to	Automatically saves a backup copy of the file either to a folder named VW Backup (in the same folder as the original file) or to a custom location, such as a network drive (click **Choose** to select a folder). The original file is not saved automatically; to save it, use one of the save commands on the **File** menu. Backup files have unique names that include the word "Backup" and a date and time stamp appended to the original file name. Use backup file(s) to restore a project if something happens to the original file.
Keep the ___ most recent backups	If **Autosave a backup copy to** is selected, specifies the maximum number of backup files to keep; the oldest backup file is replaced when a new backup is made

Click here for a video tip about this topic (Internet access required).

Interactive Preferences

Click the Interactive tab to set preferences for the display of the interactive drawing features, such as the cursor, selection boxes, and selection highlighting. See "Drawing with Snapping" on page 131 for more information about snapping. See "Selection and Pre-selection Indicators" on page 114 for more information about highlighting.

Click to show/hide the parameters.

Parameter	Description
Cursor	
Full screen cursor	Displays a crosshair cursor that extends to the edges of the drawing area
Show selection box	Displays a box beneath the cursor that indicates the active selection area; an object beneath the box can be selected when the mouse is clicked
Show snap box	Displays a box around the cursor that indicates the area in which to find possible snap points; when a snap point is beneath the box, it can be snapped to
Show acquisition hints	Show graphical hints near the cursor when smart points, edges, and vector locks can be acquired (see "Snapping Indicators" on page 144)
Selection box size	Controls the size of the selection box, if **Show selection box** is selected; must be smaller than the snap box. The actual size of the selection box is shown.
Snap box size	Controls the size of the snap box, if **Show snap box** is selected; must be larger than the selection box. The actual size of the snap box is shown.
Highlighting	
Selection highlighting	When enabled, objects that are selected are highlighted with the specified colors and patterns, and square handles indicate points that can be used to reshape objects with the tool that is currently active. When this option is disabled, square handles indicate objects that are selected, and the handles may or may not be reshape points.

Parameter	Description
Animation	If **Selection highlighting** is selected, specify how the selection highlights will be animated when the cursor moves out of the drawing window or over a palette: • Off turns off all animation of highlights • On pulses the highlight continuously • Burst briefly highlights with 100% opacity
Cursor pre-selection highlighting	Highlights any object that can be selected when the cursor is over the object
Highlighting timer	When a drawing object is beneath the cursor, specifies the number of seconds to delay before pre-selection highlighting begins
Marquee pre-selection highlighting	As a selection marquee is drawn over objects, highlights any object that will be selected
Snapped object highlighting	Highlights the geometry that generated the current snap
Tool highlighting	Highlights any object that can be modified or used by the current tool when the cursor is over the object
View transition animation	Animates the transition from the current view to a new view, providing a smoother experience when changing views. Rendered views may temporarily switch to wireframe view during the transition, specifically when switching from sketch to wireframe, dashed hidden line to interactive hidden line, and any polygon or Renderworks rendering mode to OpenGL
Interactive Appearance Settings	Opens a dialog box to change the appearance of the interactive elements in Vectorworks, including general elements, object highlighting, SmartCursor elements, and snap points; see “Configuring Interactive Display” on page 116

User Folders Preferences

You can designate which folder will hold your user data and preferences files. The subfolders within this folder mirror several of the standard subfolders in the Vectorworks application folder (Libraries, Plug-Ins, Settings, Workspaces, and so on).

This duplication of folders in separate locations makes it easy for users to do the following:

- Back up (or transfer to another computer) custom content and preferences
- Work in an environment where user permissions are limited (such as a school lab)
- Switch between users on the same computer
- Update the Vectorworks software with no loss of custom files

The Vectorworks software provides a variety of content to aid in drawing, but you can also create custom content and place it in your user folder (see “Creating Custom Default Content Libraries” on page 219). When the Vectorworks program presents content in a palette or dialog box, it includes content from your user folder as well as the content it ships with. For example, when you select **Modify > Hatch** to edit a hatch, the Hatches dialog box shows default hatches from the Vectorworks folder (where the software is installed), and your user folder.

If the same file name exists in more than one folder, only the content from one of the files is shown in the Vectorworks program, according to the following priority: user folder, application folder. For example, if the following files exist, only hatches from the file in the user folder show in the hatch list in the Attributes palette in the Vectorworks program.

- [User]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx

- [Vectorworks]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx

  If Vectorworks Design Series products are installed, you can also access custom content from shared workgroup folders on a network; see “Sharing Custom Content Using Workgroup Folders” on page 216.

Click the User Folders tab to specify the user folder.

Click to show/hide the parameters.

Parameter	Description
User Data and Preferences Folder	Specifies the folder that contains program preferences, log files, workspaces, and any personal content you create. This might be a folder on the local computer, or on a USB drive or network drive; this allows you to run Vectorworks from any computer. See “User Data and Preferences Folder” on page 58.
Choose	Click **Choose** to change the user data folder. The program must be restarted if you change the location of the user data. See “User Data and Preferences Folder” on page 58 for details.
Explore (Windows) or Reveal in Finder (Mac)	To look at the contents of the current folder, click **Explore** (to open Windows Explorer) or click **Reveal in Finder** (to open Mac Finder)
Workgroup and Project Folders (Vectorworks Design Series required)	Permits access to content for the Vectorworks program from a location that is shared with other users. To share customizations with other users, place the files in a network folder, and then notify other users so they can designate that folder as a workgroup folder. See “Sharing Custom Content Using Workgroup Folders” on page 216 for details.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resetting Vectorworks Preferences
Setting Document Preferences
Setting Quick Preferences

User Data and Preferences Folder

The User Data and Preferences folder contains the Vectorworks files that are created and used by you. Within this folder, the program automatically creates subfolders for Libraries, Plant Database (Vectorworks Landmark required), Plug-Ins, Settings, VWHelp, and Workspaces. The following customizations are automatically saved to the user folder.

- Vectorworks Preferences settings
- Workspace Editor changes
- Log files
- Resource Browser Favorite files
- SmartCursor settings
- Saved export and import options for DXF/DWG files
- Saved settings for the **Eyedropper** and **Select Similar** tools
- Settings for certain plug-ins, such as the **Stair** and **Space** tools
- Dialog box and palette positions

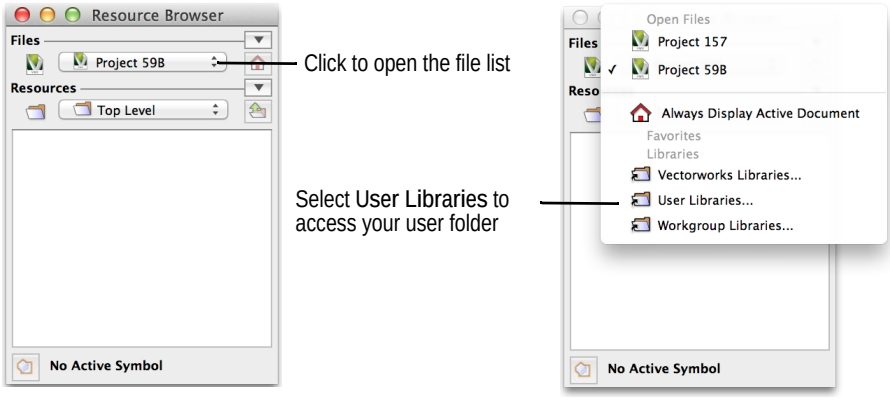
You can also manually add custom files to the Libraries folder, to have resources such as symbols and hatches display as favorites in the Resource Browser, or as default content in dialog boxes and palettes. See “Resource Libraries” on page 219 for more information.

The program defaults the following locations for the user data folder, but they can be changed. Note that, by default, Windows hides the application data folder; if you want to use the default location, adjust the folder options in Microsoft
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Explorer to make hidden folders visible. Because the user folder is outside of the Vectorworks installation folder, your data and preferences remain undisturbed when the Vectorworks program is updated.

- **Mac:** /Users/<Username>/Library/Application Support/Vectorworks/2015/
- **Windows XP:** C:\Documents and Settings\<Username>\Application Data\Nemetschek\Vectorworks\2015\
- **Windows Vista/7/8:** C:\Users\<Username>\AppData\Roaming\Nemetschek\Vectorworks\2015\

To easily access the user folder, use the file list in the Resource Browser. See "Accessing Existing Resources" on page 229 for more information.



Accessing the user folder from the Resource Browser

To change the user data folder:

1. From the User Folders tab, click **Choose**.
2. A confirmation dialog box displays. Click **Yes** to continue with the folder change.
3. Select a folder from the dialog box that opens, and click **OK** (Windows) or **Choose** (Mac).
4. Another confirmation dialog box displays. Click **Yes** to copy the user data to the new location, or click **No** to use the Vectorworks defaults.
5. If you copy the data to the new location, and the destination folder already contains a file with the same name as a file in the source folder, the program displays a notice that files in the destination folder will be overridden. Click **Yes** to continue.
6. If any unsaved files are currently open, you are prompted to save them. Click **Yes** to continue.
7. The program copies the files to the new location and then closes automatically.
8. Restart the program.

Setting Vectorworks Preferences

Resetting Vectorworks Preferences

Click the **Reset** button at the bottom of the Vectorworks Preferences dialog box to reset the preferences to their defaults; this also clears the most recently opened files list and the font mapping table. Serial number(s), user name, and company name are left intact. This operation cannot be undone.

Setting Document Preferences

Document preferences apply only to the current drawing, and they remain in effect until they are changed. When you create a template, the current document preferences are saved with the template (see “Cutting Sections” on page 1855).

To change the preference settings in the document:

1. Select **File > Document Settings > Document Preferences**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) in the drawing area, and select **Document Preferences** from the document context menu.

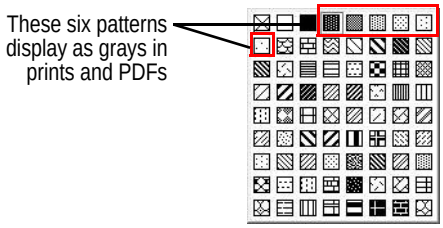
   The Document Preferences dialog box opens, with the following tabs: Display, Dimensions, Resolution, Plane Mode, and Plan Shadows (Vectorworks Architect or Landmark required for Plan Shadows).

2. Click one of the tabs to set the document preferences for that tab.

Document Display Preferences

Click the Display tab to set the display preferences.

Click to show/hide the parameters.

Parameter	Description
Black and white only	Draws objects using only the colors black and white (black items display as black, and all colors—including gray—display as white); this choice overrides any other color settings (including viewport settings) and is used mainly for printing on black and white printers. To create a “grayscale” effect for prints and PDFs when this feature is enabled, use pattern fills instead of solid color fills. Set the pattern foreground color to black, and set the background color to any other color. Patterns 4 through 9 in the pattern fill selection box display as various shades of gray. 
Hide details when layer scale < = 1:	Select to hide wall components and stair details when the layer is set to, or is below, the scale ratio specified (this setting does not affect wall component display in viewports; see “Advanced Sheet Layer Viewport Properties” on page 1642 to show or hide components in viewports)
Save viewport cache	Indicates whether to save viewport caches in the file; saving the cache may increase file size, but saves time when files that contain viewports are opened. If deselected, any viewports will require updating when the file is opened, but file size is reduced. Note that the viewport cache(s) are already compressed to PNG or JPEG format to save space (the format depends on the selection in Vectorworks preferences; see “Edit Preferences” on page 49 for more information).

Parameter	Description
Display viewport out-of-date border	Choose whether to display a red and white striped border around sheet layer viewports and design layer section viewports that do not reflect the most recent changes to a drawing. Regardless of this selection, the **Update** button's text displays in red on the Object Info palette when the viewport needs to be updated (see "Viewport Status" on page 1664).
Save site model cache (Vectorworks Architect or Landmark required)	Stores a filtered copy of the triangulated source data and the existing and proposed site model(s), so that the site model updates more efficiently. If disabled, updating the site model causes the source data to be recalculated, but file size is reduced.
Use layer colors	When specific pen and fill colors have been set for a design layer, draws all objects on that layer with the specified colors (see "Setting the Design Layer Color" on page 170)
Mesh smoothing with crease angle	Select to smooth mesh objects rendered with OpenGL or Renderworks; enter a higher crease angle value for a smoother surface (valid values are 0 to 180) To activate mesh smoothing only for selected mesh objects, see "Applying a Texture to an Object" on page 1530.
Use automatic drawing coordination (Vectorworks Design Series required)	Automatically coordinates and updates sheet numbers and drawing numbers among sheet borders, drawing labels, and section markers
Adjust flipped text	Re-orients mirrored, rotated, and flipped text in symbols, plug-in objects, and text objects so that the text is always readable
Cropped Perspective	Toggles the rectangular frame around perspective views on and off, switching between a cropped perspective view for presentation and an uncropped perspective view for modeling; see "Cropped and Uncropped Perspective Views" on page 1144

Click here for a video tip on this topic (Internet access required).

Dimension Preferences

Click the Dimensions tab to set the dimension preferences.

Click to show/hide the parameters.

Parameter	Description
Associate dimensions	Associates dimensions with the applicable objects. When a dimension is associated with an object, the dimension automatically updates when the object it is applied to is modified. For more information about associative dimensioning, see "Associative Dimensioning" on page 1191.
Auto associate	Select this option to automatically associate a dimension to the top-most object when more than one object shares a dimension endpoint
Create dimensions in dimension class	Assigns dimensions to the Dimension class as they are created (default setting). If deselected, created dimensions are assigned to the active class.

Parameter	Description
Dimension Standard	Select the default Dimension Standard to use, or click **Custom** to add a custom dimension standard (see “Using Custom Dimension Standards” on page 1187). Changing the dimension standard does not affect dimensions that have already been placed on the drawing.
Dimension Slash	Sets the desired **Thickness** of the slash at each end of a dimension, in points, mils, or millimeters

Default dimension standards are presented in the following table.

Standard	Description	Text Placement		Marker Style		Text Rotation	
		Above Dim. Line	Within Dim. Line	Slash	Arrow	Aligned	Horiz.
Arch	Architectural Standards	X		X		X	
ASME	American Society of Mechanical Engineers		X		X		X
BSI	British Standards Institute	X			X	X	
DIN	German Standards	X			X	X	
ISO	International Standards Organization	X			X	X	
JIS	Japanese Industrial Standards	X			X	X	
SIA	Swiss Standards	X		X		X	
ASME Dual Side By Side	American Society of Mechanical Engineers		X		X		X
ASME Dual Stacked	American Society of Mechanical Engineers		X		X		X

Click here for a video tip on this topic (Internet access required).

Resolution Preferences

Click the Resolution tab to set the resolution preferences.

Click to show/hide the parameters.

Parameter	Description
Rotated Text Display	Sets how rotated text is displayed: **Highest quality** shows rotated text at the best quality available, **Normal quality** shows rotated text slightly jagged, and **Bounding box** shows only a bounding box representing the text’s location
Bitmap Display	Sets how bitmaps are displayed: **Full resolution** shows bitmaps at the best resolution available, **Reduced resolution** shows bitmaps at a reduced detail, and **Bounding box** shows only a bounding box representing the bitmap’s location. Reduce the resolution to save time when using the **Pan** tool or scroll bars.
Output	

Parameter	Description
Design layer raster rendering DPI	Sets the resolution for renderings when the **Print** command is used. This setting controls only the design layer Open GL and Renderworks (if installed) rendering resolution for print; it significantly affects the output file size. Any PICT/PDF rasterization and Vectorworks geometry will print at the resolution of the printer. For bitmaps and rendered viewports, each sheet layer's DPI setting determines the upper limit for print resolution. See "Setting the Print Resolution" on page 1765 for more information.

Click here for a video tip on this topic (Internet access required).

Plane Mode Preferences

Click the Plane Mode tab to set the plane mode preference, depending on how you prefer to work and what you are drawing. Planar tools, such as the **Rectangle** tool, normally can operate in either the screen plane or in the current working plane, which is usually the layer plane. With this preference setting, planar tools can be set to operate in the screen plane only, active 3D working plane only, or any available plane, depending on the option selected. This setting does not affect tools that require an active working plane, such as the **NURBS Curve** tool, tools like the **Split** tool that always operate in the screen plane, or tools that are not affected by the planar choice, such as the **Eyedropper** tool. See "Vectorworks Modeling Environment" on page 152 for a discussion of how planes function in the Vectorworks software.

The plane mode preference does not affect the Object Info palette planes list.

Click to show/hide the parameters.

Parameter	Description
Screen Plane Only	Planar tools operate on the screen plane only; all created 2D objects are set to the screen plane. When a planar tool is active, only the screen plane can be selected from the Active Planes list on the View bar (see "The Active Planes List" on page 1168).
Working Plane Only	Planar tools operate on the active 3D plane, which is usually the layer plane. All created 2D objects are set to the active 3D plane, and when a planar tool is active, the screen plane cannot be selected from the Active Planes list on the View bar.
Screen Plane or Working Plane	When a planar tool is active, either screen plane or any of the available 3D planes can be selected from the Active Planes list on the View bar

A L Plan Shadows Preferences

In the Vectorworks Architect and Landmark products, click the Plan Shadows tab to set the shadow style preferences for plants and massing models. These preferences apply in Top/Plan view for plants that are set to use the document preference shadow settings (available only in Vectorworks Landmark; see "Plant Definition: Render Pane" on page 789) and for massing models set to display shadows (see "Creating a Massing Model" on page 718).

Click to show/hide the parameters.

Parameter	Description
Shadow Settings	
Offset	Enter a value for the shadow offset

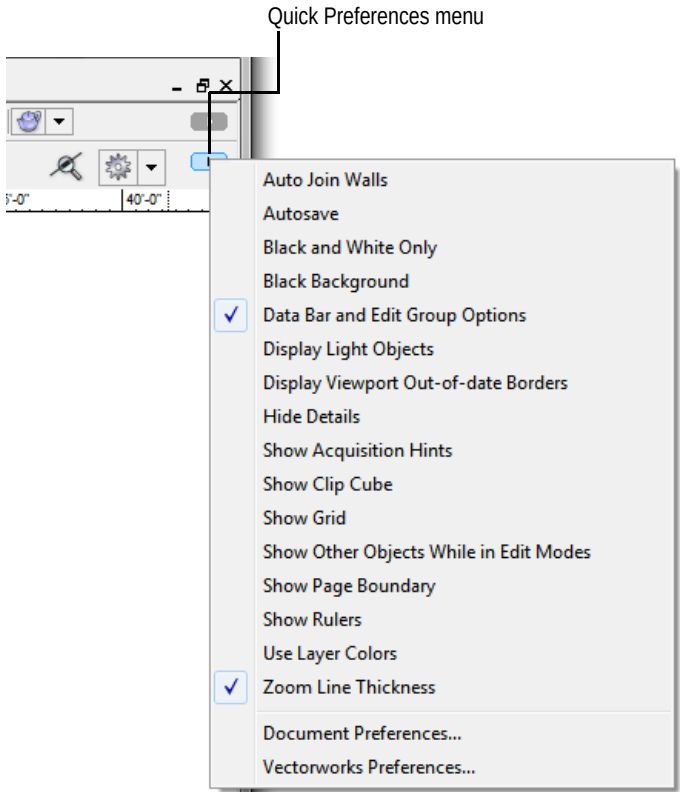
Parameter	Description
Offset Units	Select the shadow offset units. Factor of Object Height calculates the offset based on the object's height and the offset value.
Angle	Set the shadow angle by entering a value between -180° to 180° or by using the slider. 0° is straight up.
Fill Style	Select a fill style for the shadow
Color/Resource/Class	Depending on the fill style selected, select a fill color, resource (hatch, image, gradient, tile), or class for the shadow (see "Applying Object Attributes" on page 1093)
Opacity	Set the shadow opacity by entering a value between 0 and 100% or by using the slider
Use Class Opacity	Select to use the object's class opacity setting (see "Setting Class Properties" on page 179)
Top/Plan Preview	Provides a dynamic representation of the shadow values selected

Setting Vectorworks Preferences
Setting Quick Preferences

Setting Quick Preferences

The most commonly used Vectorworks, document, and interactive preferences are available for access from buttons on the Tool bar. Other commonly used settings, such as those for the clip cube, are similarly available. The **Quick Preferences** menu controls which buttons display on the Tool bar. The Data Bar and Edit Group Options list and the Zoom Line Thickness preference are enabled by default on the **Quick Preferences** menu.

Select the desired item to display its shortcut button on the Tool bar, if it is not already present. A check mark to the left of an item indicates that it is selected. Buttons display in the order the item was selected, so the first selected item displays the furthest left on the Tool bar. If the application window is resized, the Tool bar will truncate the display of buttons when necessary and replace them with ellipses on the right.



Menu Commands	Description of Button Functionality
Auto Join Walls 	Displays the **Auto Join Walls** preference button (see “Edit Preferences” on page 49)
Autosave 	Displays the **Autosave** preference button (see “Autosave Preferences” on page 55)
Black and White Only 	Displays the **Black and White Only** preference button (see “Document Display Preferences” on page 60)
Black Background 	Displays the **Black Background** preference button (see “Vectorworks Display Preferences” on page 50)
Data Bar and Edit Group Options 	Displays the Data Bar and Edit Group Options list (see “Data Bar and Edit Group Options” on page 127)
Display Light Objects 	Displays the Display Light Objects list, to specify whether lights are visible always, never, or only in Wireframe mode (see “Vectorworks Display Preferences” on page 50). The displayed icon matches the currently selected option.

Menu Commands	Description of Button Functionality
Display Viewport Out-of-date Borders 	Displays the **Display Viewport Out-of-date Borders** preference button (see “Document Display Preferences” on page 60)
Hide Details 	Displays the **Hide Details** preference button (see “Document Display Preferences” on page 60)
Show Acquisition Hints 	Displays the **Show Acquisition Hints** preference button (see “Interactive Preferences” on page 56)
Show Clip Cube 	Displays the **Show Clip Cube** button (see “Viewing a Model with the Clip Cube” on page 1155)
Show Grid 	Displays the **Show Grid** preference button (see “Grid Snapping” on page 133)
Show Other Objects While in Edit Modes 	Displays the **Show Other Objects While in Edit Modes** preference button (see “Vectorworks Display Preferences” on page 50)
Show Page Boundary 	Displays the **Show Page Boundary** preference button (see “Page Setup” on page 1761)
Show Rulers 	Displays the **Show Rulers** preference button (see “Vectorworks Display Preferences” on page 50)
Use Layer Colors 	Displays the **Use Layer Colors** preference button (see “Document Display Preferences” on page 60)
Zoom Line Thickness 	Displays the **Zoom Line Thickness** preference button (see “Vectorworks Display Preferences” on page 50)
Document Preferences	Opens the Document Preferences dialog box (see “Setting Document Preferences” on page 60)
Vectorworks Preferences	Opens the Vectorworks Preferences dialog box (see “Setting Vectorworks Preferences” on page 49)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Vectorworks Preferences
Setting Document Preferences
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Drawing Setup

Setting up the Drawing

Before beginning a new drawing, determine drawing properties like scale, units, interactive displays, color palettes, line thickness, and dimension standards. See “Palette Layout Options” on page 40 to set the preferred window options prior to setting up the drawing.

The combination of layers, classes, and views can produce a variety of drawings generated from a single file (see “Drawing Structure” on page 155). Proper file setup ensures maximum usability for multiple output.

The three required drawing settings are layer scale, units, and drawing size. Once these are established, optional drawing settings can be found in document preferences, scripts, snap and reference grids, layers and classes, and other defaults; although making these settings is not required, it is recommended.

Setting up the layers and classes while setting up the drawing is recommended. See “Layers” on page 161 and “Classes” on page 176 for more detailed information about creating layers and classes.

When using templates, many of these settings will have already been specified.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Design Layer Scale

Layer scale is the ratio of the actual size of an object to its size in a drawing. Layer scale exists to allow graphic properties of the drawing or model to be represented properly, as though you were drawing at a particular scale on a piece of paper. For example, for the default layer scale of 1:1, every inch in the drawing represents an inch in the “real world.” The scaling value allows the proper representation of attributes such as line thickness, line type, marker size, text size, and hatches.

In “What You See Is What You Get” (WYSIWYG) drawing, an intended output scale is needed to properly display these attributes; this allows you to see how the drawing will look at that intended format. For practical use, set the layer scale to be the same as that of the predominant output scale of the project. This will necessitate the least amount of attribute scaling in viewports.

Vectorworks can set one scale for all design layers in a drawing, or set a different scale for individual design layers.

Changing the Scale of the Drawing or the Active Design Layer

To change the scale of the active design layer or of the entire drawing:

1. Right-click (Windows) or Ctrl-click (Mac) in the drawing area to access the document context menu, and then select **Active Layer Scale**. Alternatively, if the layer scale is displayed on the View bar, click the **Layer Scale** button.

   The Layer Scale dialog box opens.
2. Either select a scale, or enter a custom value in **Paper Scale**.
3. To simultaneously change the scale of all existing design layers in the entire drawing, select **All Layers**.
4. To scale text proportionally to the rest of the design layer, select **Scale Text**. Deselect **Scale Text** to keep the text at its current size even if the scale changes.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

5. Click **OK** to return to the drawing.

Changing the Scale of Selected Design Layers

To change the scale of one or more selected design layers:

1. Select **Tools > Organization**.

   The Organization dialog box opens. Click on the Design Layers tab.

2. Select the layer(s) to change from the **Design Layer** list and click **Edit** (or double-click a single layer name).

   The Edit Design Layers dialog box opens (see “Setting Design Layer Properties” on page 165).

3. Click **Scale**.

   The Layer Scale dialog box opens.

4. Either select one of the scales, or enter a custom value in **Paper Scale**.

5. To scale text proportionally to the rest of the design layer, select **Scale Text**. Deselect **Scale Text** to keep the text at its current size even if the scale changes.

6. Make sure that **All Layers** is not selected.

7. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Design Layer Properties
Units

Units

The Vectorworks program provides a wide range of measurement systems to select from and also provides the flexibility to create a customized measurement system. The selected measurement system is applied globally throughout the drawing, from the measurements that display on the rulers to those used in dimensions and worksheets.

The **Units** command opens the Units dialog box, which organizes units settings on two tabbed panes:

- The General Display and Dimensions tab settings affect the units throughout the drawing and the rounding settings for primary dimensions.
- The Dual Dimensions tab contains many of the same parameters found on the General Display and Dimensions tab, but its settings apply only to secondary dimensions. If dual dimensions are not being used, these settings have no effect. For more information on dual dimensioning, see “Dual Dimensioning” on page 1200.

Selecting a Unit System

To select or change the current measurement system:

1. Select **File > Document Settings > Units**.

   The Units dialog box opens.

2. Specify the units display parameters, and then click **OK**.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Length	
Units	Select the desired length measurement system from the list. If this setting is changed from one metric unit to a different metric unit, the **Decimal precision** and **Dimension object precision** values are automatically scaled by the same ratio, to maintain the same inherent precision the document had before. If the setting is changed from a non-metric unit to a metric unit, the precision values are automatically set to new default values suitable for the selected metric unit.
Custom	If the Custom unit is selected, click **Custom** to create a custom length measurement system; see "Creating a Custom Unit System" on page 70
Show Unit Mark	Select to display the unit mark along with the unit value; if the Feet & Inches unit is selected, Vectorworks automatically displays unit marks and therefore, **Show Unit Mark** is unavailable
Show Thousands Separators in Dimension Text	Select to display a thousands separator in the values of dimension objects; the separator used (a period or comma) depends on the current regional setting in the operating system Separators are for display only; they are not used in the Object Info palette, or in the dimension text as it is being edited
Rounding Style	Rounding only affects how numbers are displayed; if the number 1.23456 is entered with a rounding of .00, the value is recognized as 1.23456 but displays as 1.23
Fractional	Select for fractional rounding
Decimal	Select for decimal rounding
Exact as Fractions / Non-Exact as Decimals	Select for a combination of fractional and decimal rounding
Fractional Display for Dimensions	If a fractional **Rounding Style** was selected, select a style for the fractions that appear in dimensions: • Standard: 2 1/4 • Diagonally stacked: 2 1⁄4 • Vertically stacked: $2\frac{1}{4}$
Rounding Precision	
Fraction precision	If a fractional or combination **Rounding Style** was selected, select the fractional precision value, up to 1/64
Decimal precision	If a decimal or combination **Rounding Style** was selected, select up to ten digits of decimal precision
Dimension object precision	Select a precision value for dimension objects; to link the dimension value to the fraction or decimal precision value, click the link button to the right of the precision fields: 
Decimal rounding base	Select whether decimal rounding is performed using multiples of tenths, quarters, or halves; the option selected is reflected in the **Decimal precision** field

Parameter	Description
Dimension rounding base	Select whether dimension rounding is performed using multiples of tenths, quarters, or halves; to link the dimension value to the decimal base value, click the link button to the right of the rounding base fields: 
Decimal Options	
Leading Zero	If one of the decimal rounding options is chosen, select to display a leading zero
Trailing Zeros	If one of the decimal rounding options is chosen, select to display trailing zero(s)
Area / Volume / Angle	
Units	Select the area, volume, and angular measurement system from the list. If this setting is changed from one metric unit to a different metric unit, the **Precision** value is automatically scaled by the same ratio, to maintain the same inherent precision the document had before. If the setting is changed from a non-metric unit to a metric unit, the precision value is automatically set to a new default value suitable for the selected metric unit.
Custom	If the Custom area or volume unit is selected, click **Custom** to create a custom area or volume measurement system; see “Creating a Custom Unit System” on page 70
Precision	For area and volume measurement systems, select up to ten digits of decimal precision. For angular measurement systems, select to display angular units in degrees (there are 360 degrees in a circle, and the angle mark is °), radians (there are 2pi radians in a circle, and the angle mark is r), or gradians (there are 400 gradians in a circle, and the angle mark is g); also, select up to eight digits of angular precision or specify degrees, minutes, and/or seconds.

Creating a Custom Unit System

To create a customized measurement system:

1. Select **File > Document Settings > Units**.

   The Units dialog box opens.

2. Select **Custom** from the length, area, or volume **Units** list, or if editing an existing custom measurement system, select the custom name from the **Units** list.

3. Click **Custom**.

   The Custom Units:Length, Custom Units:Area, or Custom Units:Volume dialog box opens, depending on the type of custom unit being created or edited. The available values in each field vary based on the custom unit type.

Click to show/hide the parameters.

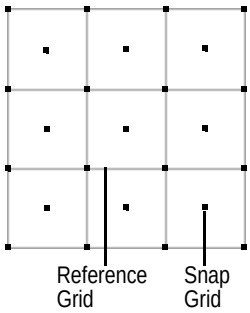
Parameter	Description
Unit Name	Name given to the custom measurement system—for example, “Cubit”
Unit Mark	Mark (abbreviation) used to represent a unit in the custom measurement system—for example, “cbt”
Smaller from Larger	Set the number of custom units that occur per selected unit—for example, “**There Are** 2 **Units Per** Inch”

Parameter	Description
Larger from Smaller	Set the number of selected units that occur per custom unit—for example, "**There Are** 2 Inches **Per Unit**"

4. Click **OK** to exit the Custom Units dialog box.
5. Enter the remaining criteria as described in "Units" on page 68.
6. Click **OK**.

Snap and Reference Grids

There are two separate grid systems for precise drawing: the snap grid and the reference grid.



The **Snap Grid** assists with drawing and placing objects precisely. The snap grid appears as small dots when zoomed in, depending on the snap grid dimensions. The snap grid is used in combination with the **Snap to Grid** snapping option on the Snapping palette. As the mouse moves across the screen, it automatically "catches" at each increment on the snap grid. When you are placing or drawing an object, it snaps to the grid.

With shortcut keys, objects can be moved constrained to the snap grid. See "Edit Preferences" on page 49 for more information.

The **Reference Grid** normally displays on the screen (depending on the zoom factor and grid dimensions), and can be printed. The most useful way to set the reference grid is so that it is an extension of the set scale.

Depending on the drawing, the snap and reference grids can have identical or different dimensions. For example, if you are designing kitchen cabinets with a tolerance of one-sixteenth of an inch, set the snap grid to 1/16". It would be hard to draw on a screen that displayed 16 horizontal and vertical lines squeezed within every inch. For that reason, you might want to mark off only whole inches on the screen by setting the reference grid to 1".

At file setup, establish the snap and reference grid settings based on the drawing scale. See "Grid Snapping" on page 133. The color of the reference grid can be changed; see "Configuring Interactive Display" on page 116.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Grid Snapping
Configuring Interactive Display

Guides

The **Make Guide** command enables the placement of guidelines which can be used to visually align objects in the drawing. Guides can be any shape and size. In addition to providing visual clues, guides work with the SmartCursor to ensure precise alignment. Guide objects are locked into a class called "Guides" and are colored light purple. Guides are printed unless the **Hide Guides** command is selected, or the **Delete All Guides** command has permanently removed them.

To create a guide:

1. Create the object to become a guide.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Select the object.
3. Select **Modify > Guides > Make Guide**.

   The object changes into a guide and is placed in a Guides class.

The following commands can be selected from the **Modify > Guides** menu.

Option	Description
Make Guides	The object becomes a guide
Select Guides	Selects all the guides in a drawing
Show Guides	Displays guides which were hidden
Hide Guides	Temporarily hides the guides in the drawing
Delete all Guides	Permanently removes all guides and the objects used to create them

You can snap objects to the guide(s) closest to it by using **Snap to Object** from the Snapping palette. The distance at which the objects snap to the guides is determined by the **Snap box size** setting in the Vectorworks preferences. See “Interactive Preferences” on page 56 for details on setting the snap radius setting.

To delete a single guide, highlight it and then select **Modify > Unlock**. The guide is now editable. Select **Edit > Clear** to remove it from the drawing. A guide can also be unlocked in order to move it to a new location. Select **Modify > Lock** to lock the guide into place once it is relocated.

Internal Origin and User Origin

The center of a Vectorworks drawing has fixed coordinates of (0,0); this location is called the internal origin. A user origin also exists. Drawing coordinates display relative to the user origin. Normally, the internal origin, the user origin, and the page center are all coincident. While the internal origin cannot move, the user origin and page center can be moved.

Depending on the design needs, and for some imported files (to maintain the imported file’s coordinates), the user origin needs to shift so that the user origin location is different from the internal origin. One reason to change the origin is to make it easier to work with coordinates, measurements, and distance values. For example, the Swiss use a reference point system for architectural measurements, where everything is in relation to a point in Europe. When a building is located at a site, that site is referenced to be a certain distance and direction from this point. Using this system creates large numbers on the ruler and in the Object Info palette. In the User Origin dialog box, you can shift the user origin close to the building site so that the system does not display large coordinates.

Changing the user origin can have far-reaching consequences. Use caution when changing the user origin.

Several commands related to the user origin and the internal origin are available in the Vectorworks program.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Locating the Internal Origin
Centering the Drawing on the Internal Origin
Setting the User Origin

Locating the Internal Origin

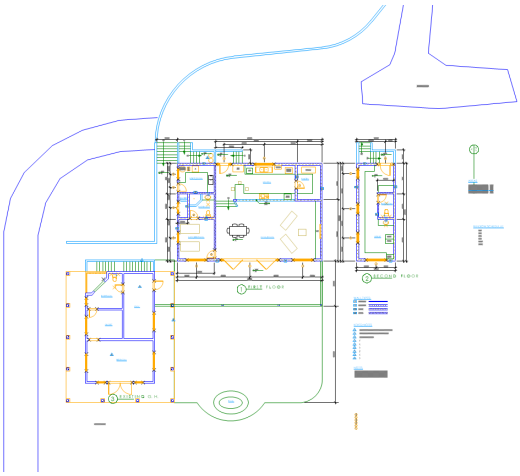
In some drawings, objects may be located several kilometers or miles away from each other and/or from the drawing’s internal origin. For reference, it is easy to change the view to locate the internal origin with the **Locate Internal Origin** command.

To locate the internal origin relative to visible objects:

1. Select **Tools > Origin > Locate Internal Origin**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. The drawing view changes; it switches to Top/Plan view if needed, and centers the internal origin in the view. The zoom factor is adjusted so that currently visible objects remain on the screen relative to the internal origin. If the internal origin marker was not set to display in the Vectorworks preferences, it is automatically turned on.

   If the objects are too far away from the internal origin, use **Center Drawing on Internal Origin** to resolve the problem.



While working on a drawing, it becomes necessary to know where the internal origin is located

The Locate Internal Origin command zooms out and centers the view on the internal origin, which displays as a marker. It becomes clear that the objects are located far away from the internal origin.

Centering the Drawing on the Internal Origin
Setting the User Origin
Internal Origin and User Origin

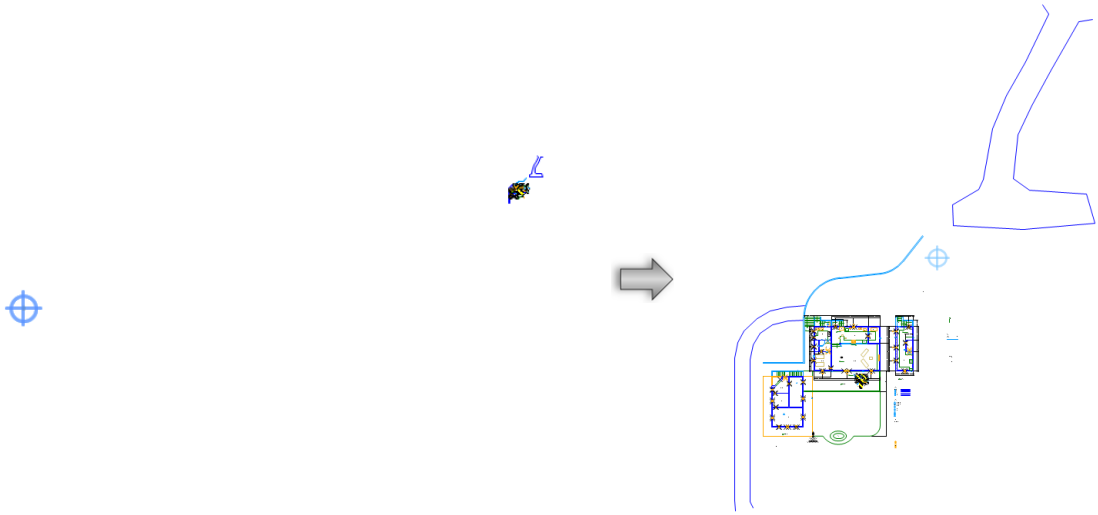
Centering the Drawing on the Internal Origin

When a drawing contains objects that are located far away from the internal origin, this can cause problems with OpenGL rendering and with calculation precision due to rounding errors. By centering the objects about the internal origin, the problems are resolved because the distance from the internal origin to the objects has been reduced. The centering operation adjusts the user origin so that the coordinates of any objects in the drawing remain unchanged.

This command moves objects in the drawing, even though the coordinates do not appear to change. The user origin changes as a result of the command to keep the coordinates unchanged.

To center the drawing on the internal origin:

1. Select **Tools > Origin > Center Drawing on Internal Origin**.
2. An alert displays, explaining how far away the most distant object is located from the internal origin (in document units) before the centering operation, and showing where it will be located after the operation. Click **Yes** to continue.
3. The entire drawing is centered about the internal origin, moving all objects on all design layers, including locked objects and non-visible objects. Any sheet layer viewports and saved views are automatically adjusted. Because the user origin is also adjusted, the coordinates of the file remain intact.



Objects in the drawing are located too far away from the internal origin

The Center Drawing on Internal Origin command centers the objects in the drawing about the internal origin. Object coordinates remain intact.

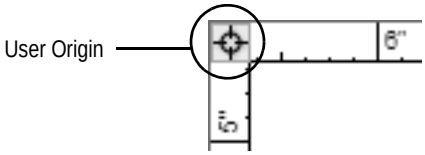
Locating the Internal Origin
Setting the User Origin
Internal Origin and User Origin

Setting the User Origin

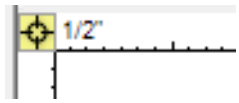
The **User Origin** command and the **User Origin** button can be used to change the location of the user origin in the drawing area. Normally, the user origin is located coincident with the internal origin, where the X and Y axes meet (0,0). Depending on design needs, the user origin may need to shift relative to the internal origin. The drawing coordinates display relative to the user origin.

All design layers have the same user origin, but each sheet layer has its own user origin.

When the user origin is different from the internal origin, the **User Origin** button displays in yellow.



The user origin and the internal origin are coincident



The user origin has been set to a location that is different from (0, 0)

To set the location of the user origin:

1. To set the user origin for a specific sheet layer, activate the sheet layer first; for design layers, switch the active design layer to Top/Plan view if not in 2D view.
2. Select **Tools > Origin > User Origin**. The User Origin dialog box opens.

   Alternatively, click the **User Origin** button once to set the user origin by the next mouse click, if this functionality has not been disabled in the User Origin dialog box. Double-click the **User Origin** button to open the User Origin dialog box.

Click to show/hide the parameters.

Parameter	Description
Set User Origin to next mouse click	Changes the cursor to a bull's-eye cursor; click to set the user origin to any point within the drawing area
Set User Origin to Internal Origin	Sets the user origin to the internal origin, making them both coincident at (0,0)
Set User Origin (relative to the Internal Origin) to	Enter the specific coordinates of the user origin relative to the internal origin. The last non-zero values of the user origin are saved; the values remain in the dialog box to be easily reselected. In this way, you can toggle between setting the user origin to a specific location and setting it to (0,0).
Next mouse click is	Sets the user origin relative to the coordinates of the next point clicked by the mouse. Sets the clicked point to specific coordinates. After selecting this option, enter the X and Y coordinates of the point. The user origin is set relative to the values entered.
Disable dragging for the Set User Origin button	Disables the **User Origin** button, so that clicking on it once does not activate the ability to set the user origin to the next mouse click. Double-clicking the button still opens the User Origin dialog box.

3. Click **OK**.

   If setting the user origin by mouse click, click to set the location of the user origin (**Set User Origin to next mouse click**) or to set the location of the offset user origin (**Next mouse click is**).

   The user origin moves to its new location. Coordinates are adjusted relative to the user origin. If a working plane is in use, the working plane coordinates are set to be relative to the user origin.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Locating the Internal Origin
Centering the Drawing on the Internal Origin
Internal Origin and User Origin

Setting Default Object Attributes

The default attributes for a document are the attributes that are automatically applied to each new object when it is created. To set the default attributes, simply adjust the Attributes palette settings with no objects selected.

When you first open a new document, set the default attributes to those you use most often. For example, you might adjust the fill and line colors, and line thickness. The available attributes can be customized as needed.

The default attributes apply to the current document only. To save a set of default attributes or custom attributes for future use, set the attributes as desired and save the document as a template.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Attributes Palette
Creating Templates

Creating Templates

Save a drawing file as a template to use it as a foundation for new files. Templates save layers, classes, title blocks, sheet borders, resources, and the current settings for attributes and units. (See "Creating a New File" on page 22 for details about using a template.)

When a template is opened, the Vectorworks program automatically opens a copy of the file. When the new drawing is saved the first time, the program prompts for a new file name. This makes it impossible to accidentally replace the master template with the new drawing file.

To create a template:

1. Start with a new, empty file.
2. Set up the file with all of the desired elements.
3. Select **File > Save As Template**.
4. Enter the name of the template (.sta) file and place it in an appropriate Templates folder. (See "User Folders Preferences" on page 57 for details about how to specify the folder for your user data.)
   - To save the template for personal use, place it in the Templates subfolder in your user data folder: [User]\Libraries\Defaults\Templates.
   - If Vectorworks Design Series products are installed, you can share the template with other users in a workgroup. To do so, place it in the Templates subfolder of a workgroup folder on a network drive; other users can then specify that workgroup folder in their Vectorworks preferences: [Workgroup]\Libraries\Defaults\Templates.

   The .sta extension is required for Windows. It is recommended for use on the Mac if the file will be shared with Windows users.
5. Click **Save**.

The program launches with a blank drawing file, in 1:1 scale. To launch the program with a file containing different settings, name the template file Default.sta and save it in the Templates subfolder in your user data folder: [User]\Libraries\Defaults\Templates.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Saving a File

AL Document Setup

Use the **Document Setup** command to set up a file's basic characteristics (units, scale, drawing area, and grid) as well as define the sheet border and title block settings.

To set up a file with the **Document Setup** command:

1. Select **File > Document Settings > Document Setup**.
2. The Document Setup dialog box opens. Set the parameters for the drawing. See "Setting up the Drawing" on page 67 for more information on units, scale, drawing grids, and print area.

Click to show/hide the parameters.

Parameter	Description
Units, Scale and Grids	
Drawing Units	Click **Change** to open the Units dialog box; specify the global unit settings for the project
Layer Scale	Click **Change** to open the Layer Scale dialog box. Specify the default layer scale for the project. This scale will be used for all floor design layers (Mod-Floor-#) and associated design layers such as Mod-Slab.
Drawing Area	Click **Change** to open the Page Setup dialog box; specify the drawing's printable area
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Drawing Grids	Click **Change** to open the SmartCursor Settings dialog box; specify the reference and snap grid settings
Georeferencing	If this document will use georeferencing, click **Change** to open the Document Georeferencing dialog box. Specify the default georeferencing information for all design layers; see "GIS and Georeferencing" on page 775.
Sheet Border/Title Block	If not created during setup, sheet borders and title blocks can be added manually with the **Sheet Border** tool (see "Adding a Sheet Border" on page 77)
Sheet Border	Select the sheet border to automatically include as the file is set up, or leave the default selection of None to create no sheet border
Title Block	Select the title block from either the default content or the current file's content to automatically include as the file is set up (see "Resource Libraries" on page 219), or leave the default selection of None to create no title block
Use Border Settings	
As preferences	Saves the sheet border and title block selections and applies them to the sheet layers when the **Create Standard Viewports** command is selected (Vectorworks Architect required)
To create a border now on layer	Places the specified sheet border and title block on the selected layer immediately

3. Click **OK**.

To use the Issue Manager (see "The Issue Manager" on page 90), select one of the predefined sheet border styles.

Select the **Document Setup** command again to re-adjust a sheet border if the paper size has changed.

Adding a Sheet Border

All Vectorworks products include pre-formatted sheet borders for project viewports or views; they can be added to a file in various ways.

The **Sheet Border** tool places a pre-formatted border along the edges of the drawing area, set to the drawing size. Standard size pages have a matching standard sheet border. A custom size border can easily be specified, and sheet borders and title blocks can be customized to meet office requirements.

The sheet border in the Vectorworks Design Series products has expanded capabilities. Additional default content and parameters are available only in the Vectorworks Design Series products. Various types of sheet borders can be created, including ConDoc, GSA, AEC, ASME, and ISO formats and their associated title blocks, revision blocks, tolerance blocks, projection blocks, and issue and revision data.

To work with the Issue Manager, special considerations apply when the title block fields are formatted. See "Creating a Custom Title Block" on page 86.

The **Update Plug-in Objects** command may need to be run on files that contain sheet borders (formerly known as "drawing borders") that were created in an earlier version of the Vectorworks program. This command converts the drawing borders to the latest format; see "Migrating from Previous Versions" on page 28. Additional sheet border capabilities are available in the Vectorworks Design Series products. See "Adding a Sheet Border" on page 77.

To place a sheet border:

1. Make the design or sheet layer active.
2. Click the **Sheet Border** tool from the Dims/Notes tool set.

   Alternatively, the Vectorworks Architect and Landmark products allow sheet borders to be set up using drawing standards. See “Document Setup” on page 76 and “Standard Viewports” on page 186.
3. Click **Preferences** from the Tool bar to set the default sheet border parameters.

   The Sheet Border Preferences dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Sheet Size	Select a standard size and format for the sheet border, or select Fit to Page to set the sheet border to the page dimensions (see “Page Setup” on page 1761). Select Custom to use custom sheet border dimensions, and specify the **Width** and **Height** dimensions.
Title Block	Opens the Import Title Block dialog box, for selecting a title block symbol to insert, or select None for no title block
Lock to Page Center	Locks the sheet border center position to the page center; deselect to position the sheet border manually. If the plan has been rotated (Vectorworks Design Series required), select **Lock to Page Center** to position the sheet border correctly when in a non-rotated view.
Use As Title Block Only	When a title block has been selected, displays only the title block and hides all other sheet border elements

4. Click once in the drawing to set the sheet border insertion point, and then click again to set the sheet border orientation.
5. The sheet border is placed on the drawing. Sheet borders should be placed as 2D screen objects (See “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152).
6. Set the text attributes as needed using the **Text > Format Text** command. You can also drag and drop a text style from the Resource Browser onto the sheet border object.
7. Set the line attributes as needed using the Attributes palette.
8. The sheet border can be re-sized and re-scaled after placement, and title blocks and revision histories can be added.

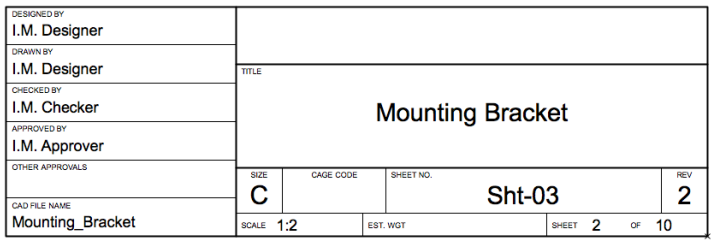
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

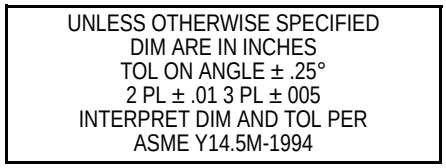
The Attributes Palette
Formatting Text
Using Text Styles
Sheet Border Properties
Adding a Title Block
Creating a Custom Title Block

Sheet Border Properties

The sheet border properties can be edited in the Object Info palette.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Plane	Generally, for sheet borders, select Screen to place the border on the screen plane. See “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152.
Lock to Page Center	Locks the sheet border center position to the page center; deselect to position the sheet border manually. If the plan has been rotated (Vectorworks Design Series required), select **Lock to Page Center** to position the sheet border correctly when in a non-rotated view.
Size	Select a standard size and format for the sheet border, or select Fit to Page to set the sheet border to the page dimensions. Select Custom to use custom sheet border dimensions, and click **Border Settings** to specify the dimensions.
Orientation	Select Portrait or Landscape orientation
Horizontal Dimension	Displays the sheet border horizontal dimensions
Vertical Dimension	Displays the sheet border vertical dimensions
Border Settings	Opens the Sheet Border Settings dialog box, for specifying further properties (see “Specifying Additional Sheet Border Settings” on page 81)
Title Block	Opens the Import Title Block dialog box, to select a title block symbol to insert (see “Adding a Title Block” on page 83). This dialog box can also be accessed from the sheet border context menu; right-click (Windows) or Ctrl-click (Mac) on the sheet border, and select **Title Block**. ASME title blocks are required to insert a tolerance or projection block.  Additional default title blocks are available with the Vectorworks Design Series products; for US Arch title blocks, information is linked to the Issue Manager. To remove a title block, select the Defaults symbol folder, and then select **None** from the **Symbols** list.
Current Title Block	When a title block has been inserted, displays the title block symbol name
Use As Title Block Only	When a title block has been selected, displays only the title block and hides all other sheet border elements
Title Block Position	Specifies the title block location relative to the sheet border
Title Blk Scale Factor	If the title block is too large or small at normal scale (scale factor 1), scales the title block size, including text. A value below 1 makes the title block smaller, while a value above 1 makes the title block larger; text is automatically scaled along with the title block geometry.
Title Blk Margin	Adds a horizontal, vertical, or block margin to the title block

Parameter	Description
Use Revision Block (Vectorworks Design Series required)	Adds a revision history block to the sheet border
Rev. Block Position	Specifies the revision block location relative to the sheet border
Rev. Block Height	Specifies the height of the revision block
Rev. Block Width	Specifies the width of the revision block
Show Revision Zone	Adds a revision zone column to the revision block, for specifying the location of the revision
Use Tolerance Block (Vectorworks Design Series required)	For ASME sheet borders, adds a tolerance specifications block to the title block. The tolerance block creates a legend containing the drawing tolerances, including both angular and linear accuracy.  UNLESS OTHERWISE SPECIFIED DIM ARE IN INCHES TOL ON ANGLE ± .25° 2 PL ± .01 3 PL ± 005 INTERPRET DIM AND TOL PER ASME Y14.5M-1994 Tolerance block details are text objects, and do not affect the actual dimensional tolerances.
Use Projection Block (Vectorworks Design Series required)	For ASME title blocks, adds a projection block area to the title block; specify **First Angle** or **Third Angle** in Projection
Show Grids	Displays grid text and lines in the sheet border margin. This setting can also be accessed from the sheet border context menu: right-click (Windows) or Ctrl-click (Mac) on the sheet border, and select **Show Grids**.
Show Grid Lines	Displays grid lines on the drawing
Fold Marks	Select the fold mark measurements when adding fold marks to the sheet border. The first measurement specifies the drawing fold width and the second measurement indicates the margin fold width. Select Custom to specify custom fold mark distances. Fold marks are designed for use with ISO drawings.
Margin Width	When custom fold marks are selected, enter the margin fold width
Folded Width	When custom fold marks are selected, enter the drawing fold width
Folded Height	When custom fold marks are selected, enter the drawing fold height
Hide Border	When the sheet border includes a title block, hides the sheet border and displays only the title block (this allows a different sheet border to be used with that title block, if desired)
Fill Border	Fills the area between the outer and inner border lines with a fill selected from the Attributes palette
Add Parts List (Vectorworks Design Series required)	Adds a parts list to the title block (this is intended for ASME title blocks, but can be added to any title block). Information from detail bubbles populates the parts list; see “Creating Detail Bubbles” on page 1274. If a parts list worksheet does not yet exist, it is created.

Parameter	Description
Edit Title Block (extended capabilities in Vectorworks Design Series)	Opens the Edit Title Block dialog box, for specifying the title block information. Depending on the selected title block, different fields and tabs are available. In Vectorworks Design Series products, ASME title blocks include a tolerance tab, for editing the tolerance block information (see “Editing a Tolerance Block” on page 85). US Arch title blocks include Project and Sheet tabs; information is entered from the Issue Manager but can be edited manually (see “The Issue Manager” on page 90). Any title block that contains a Project tab can have the same Project tab information applied to all title blocks in the file by selecting **Apply these values to all title blocks**. If **Use Automatic Drawing Coordination** is enabled in document preferences, the sheet layer is automatically updated if the **Sht Title Line 1** is changed. Similarly, if the **Sheet Number** value is changed, the sheet layer and any annotation objects (drawing labels or section markers) on the sheet layer that show this information are updated automatically.
Edit Revision Data (Vectorworks Design Series required)	Opens the Edit Revision Data dialog box, for specifying revision information and format
Edit Issue Data (Vectorworks Design Series required)	Opens the Edit Issue Data dialog box, for specifying issue data on US Arch title blocks

Specifying Additional Sheet Border Settings

Additional sheet border settings are available from the Object Info palette.

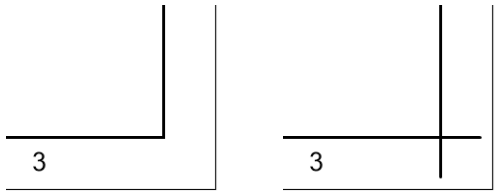
To specify additional sheet border settings:

1. Select the sheet border.
2. In the Object Info palette, click **Border Settings**. Alternatively, double-click on the sheet border.

   The Sheet Border Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Drawing Size	Specifies the sheet border size
Vertical/Horizontal Dimension	Specifies the sheet border vertical/horizontal dimensions; initially, these are based on the **Size** selected in the Object Info palette, but the dimensions can be edited. This parameter is not available when the **Size** is Fit to Page.
Dimensions Shown Are	Applies the dimensions to either the outer or inner border dimensions
Margins	Specifies the sheet border margin widths
Zones/Grids	
Vertical Zones	Specifies the number of vertical zones to include
Grid Text Order	Indicates whether vertical grid text starts at the top or bottom of the border; select Not Shown to remove grid text
Horizontal Zones	Specifies the number of horizontal zones to include

Parameter	Description
Grid Number Order	Indicates whether horizontal grid numbers start at the left or right of the border; select Not Shown to remove grid numbers
Grid Label Size	Specifies the text size for both grid text and numbers
Grid Marker Line Length	Specify the length of the grid marker lines and grid line extensions; lines cannot extend past the margin area
Grid Marker Line Weight	Select the line weight of the grid marker lines and extensions
Show Grid Line Extensions	Includes lines that extend past the border to the ends of the margin in area  Without grid line extensions With grid line extensions
Countersignature	Select whether to include a countersignature area in the sheet border margin area, and, if included, where to place it. • Select None to omit the countersignature field • Vertical places the countersignature vertically in the upper left corner of the sheet border • Horizontal places the countersignature horizontally in the upper right corner of the sheet border Sheet border margins may need adjustments to allow enough space for the countersignature if one is selected.
Reset to Default Values	If the sheet border **Size** is set to one of the standard sizes (such as US Arch B or ISO AS), select this button to restore the default sheet border settings

3. Click **OK** to set the sheet border parameters.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding a Title Block
Creating a Custom Title Block
Adding a Sheet Border
Adding a Sheet Border

Sheet List Indexing

The **Create Sheet List** command compiles the current sheet border information for use as a sheet list index or worksheet.

To create a sheet list index:

1. Select **Tools > Reports > Create Sheet List**.

   The Create Sheet List dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
US Arch Title Block Entries	Lists the current US Arch title blocks
Sheet No.	Displays the contents of the current title block selection
Add >	Moves a selected item from the US Arch Title Block Entries list to the Proposed Sheet list
< Remove	Removes a selected item from the Proposed Sheet list and displays it in the US Arch Title Block Entries list
New	Click to open the New List Entry dialog box and add the drawing and title block information to the Proposed Sheet list
Move Up/Move Dn	Moves the selected title up or down in the Proposed Sheet list
Format	Click to open the Sheet List Index Preferences dialog box and specify whether to create the sheet index as a text block or worksheet; also specify formatting criteria
Proposed Sheet List	Lists the proposed sheet borders to include in the sheet list
Sheet No./Title	Displays the contents of the current selection for editing

2. Move the desired title block entries to the proposed sheet list to create the sheet list, and format the list by clicking **Format**; select whether to create a text object or worksheet from the list.
3. Click **OK**.
4. If the sheet list index is formatted as a text block, click to select the top left and bottom right corners of the sheet list area; the text wraps to fit within this width. If the sheet list index is formatted as a worksheet, click to add the worksheet to the file.

The sheet list index worksheet can also be added to the drawing from the **VA Create Schedule** command or the Resource Browser. From the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder that is included with the Vectorworks Architect product (see “Resource Libraries” on page 219). Drag the Sheet List worksheet to the drawing. The worksheet is populated with information from the objects in the current drawing.

Adding a Title Block

A title block containing drawing information can be added to the sheet border. Title blocks are saved as symbols with text linked to a record.

A title block can be the only part of the sheet border that displays, by selecting **Use As Title Block Only** in the sheet border preferences or the Object Info palette.

Additional title block capabilities are described in “Adding a Sheet Border” on page 77.

To add a title block to the sheet border:

1. Select the sheet border.
2. In the Object Info palette, click **Title Block**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the sheet border and select **Title Block** from the context menu.

   The Import Title Block dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Symbol Folders	Specifies the location of the sheet border symbols
Symbols	Provides a graphical list of available sheet border symbols

3. Select a title block symbol from the selected symbol folder. The Defaults folder contains the default title block resources; see “Resource Libraries” on page 219. The title blocks listed under Top Level are title block symbols that exist in the current file.

   To remove an existing title block, select **None**.

4. Click **OK** to add the selected title block symbol to the sheet border. The title block is scaled to match the current layer scale if necessary.

   When a sheet border with a title block is inserted into a drawing, the Sheet Border Components symbol folder is automatically created and displays in the Resource Browser. Title block symbols added to the sheet border are automatically placed in that folder.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Custom Title Block
Creating a Custom Title Block
Sheet Border Properties

D Editing Revision Block Data

To add a revision block to the sheet border, select **Use Revision Block** from the Object Info palette of a selected sheet border. The revision block parameters (such as height and width) are specified from the Object Info palette, but the actual revisions are specified in a separate dialog box.

To add or edit revision block data:

1. From the Object Info palette of a selected sheet border with a revision block, click **Edit Revision Data**.
2. The Edit Revision Data dialog box opens.

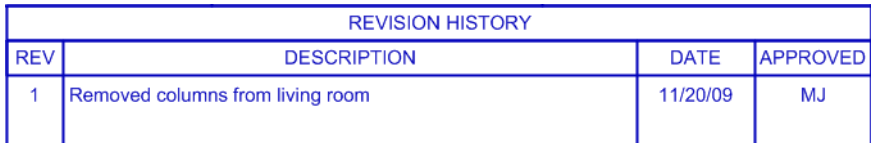
Click to show/hide the parameters.

Parameter	Description
Revisions list	Lists the current revisions displayed in the revision block
Add	Opens the Add New Revision dialog box, for adding a revision to the block
Edit	Opens the Edit Existing Revision dialog box, for editing the currently selected revision
Remove	Deletes the currently selected revision
Use	Select Letters or Numbers for the Rev# format
Show the	Specifies how many revisions to display in the revision block, from recent to oldest

3. Click **Add** to create a revision entry.

   The Add New Revision dialog box opens. Specify the revision data.

4. Click **OK** to return to the Edit Revision Data dialog box. Existing revisions can also be edited or deleted.
5. Click **OK**. The revision block is updated.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



REVISION HISTORY			
REV	DESCRIPTION	DATE	APPROVED
1	Removed columns from living room	11/20/09	MJ

Sheet Border Properties

Editing a Tolerance Block

When an ASME title block is added to a sheet border, an area for tolerance specifications data is included. The tolerance data is specified in a separate dialog box.

To edit the tolerance block data:

1. Select the sheet border from the drawing area.
2. In the Object Info palette, click **Edit Title Block**.

   The Edit Title Block dialog box opens. Click the Tolerance tab.
3. Enter the angular and dimensional tolerances to be displayed in the tolerance block.

   Only the angular accuracy, two place, and three place parameters apply to the ASME tolerance block. The remaining tolerances might apply to a custom title block.
4. Click **OK**.

   The tolerance block is updated with the new information.

Sheet Border Properties

Editing Issue Data

When a US Arch title block is added to a sheet border, an area for issue data is included. The issue notes are specified in a separate dialog box.

To add or edit issue data:

1. From the Object Info palette of a selected sheet border with a revision block, click **Edit Issue Data**.
2. The Edit Issue Data dialog box opens.

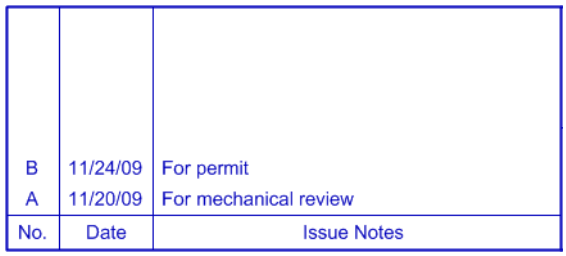
Click to show/hide the parameters.

Parameter	Description
Issue list	Lists the current issues displayed in the title block
Add	Opens the Add New Issue dialog box, for adding an issue note
Edit	Opens the Edit Existing Issue dialog box, for editing the currently selected issue note
Remove	Deletes the currently selected issue note
Use	Select Letters or Numbers for the Issue# format
Show the	Specifies how many issues to display in the title block, from recent to oldest

3. Click **Add** to create an issue entry.

   The Add New Issue dialog box opens. Specify the issue note.

4. Click **OK** to return to the Edit Issue Data dialog box. Existing issue notes can also be edited or deleted.
5. Click **OK**. The issue data is updated.



Sheet Border Properties

Creating a Custom Title Block

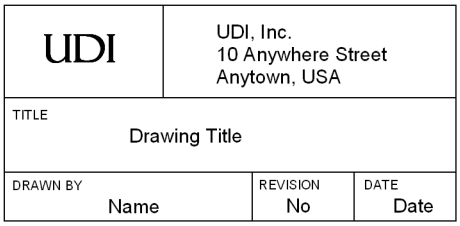
A custom title block, containing company-specific graphics, information, and data fields, can be created and then inserted into a sheet border.

Three steps are required: create a title block symbol, create a custom record format, and link the record format fields to the title block symbol text fields.

> Creating a custom title block in the Vectorworks Design Series products requires additional steps, if the custom title block is to be used with the Issue Manager.

To create a custom title block:

1. In a new file with a scale of 1:1, create the elements of the title block, including lines, rectangles, graphics, and text. Creating the title block at a scale of 1:1 ensures that it is inserted at the correct scale when it is placed on a sheet layer or a design layer of any scale. The title block should be a 2D screen object (See “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152.)



   - Do not use any 3D elements.
   - Company logos can be imported as bitmaps.
   - The title block geometry (pen color, pen style, and line thickness) and text (font, style, and color) can either inherit the sheet border attributes, or retain those attributes as created. The “By Class” setting in the Attributes palette indicates that the attributes should be inherited from the sheet border. If an attribute is not set to “By Class,” then its original setting is retained.

Description	Method
The attributes of the title block elements are inherited from the sheet border	Specify “By Class” in the Attributes palette for these elements. For text font and style attributes, set the Pen Style by class to use the same font and style as the sheet border. For text color, set the Pen Color to “Color By Class.”

Description	Method
The title block elements retain the attributes as created	Set the attributes of the geometry and text from the Attributes palette when the title block is created. For text font and style attributes, set the Pen Style to Solid to use the text attributes set when the title block was created. For text color, select the desired Pen Color.

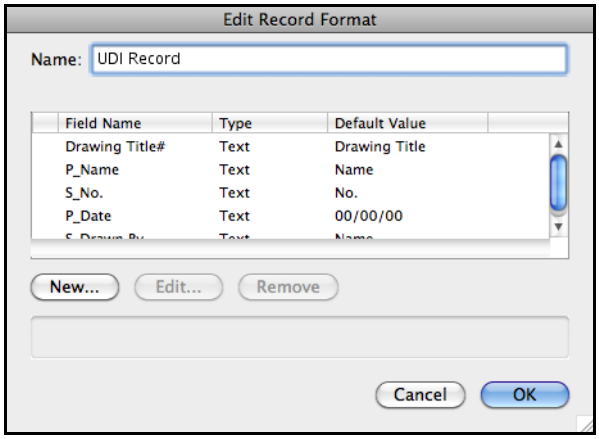
Different settings can be combined so that some attributes are inherited from the sheet border, while others remain as created. For example, if a line should use the same color as the sheet border, but have a dashed line type with a thickness of 1 mm, when creating the line, set the line's Pen Color to "Color By Class," its Line Thickness to 1 mm, and its Pen Style to the desired dashed line type.

2. Select all the title block elements, and then select **Modify > Create Symbol**.

   The insertion point of the symbol should be at lower right corner of the title block. See "Creating New Symbols" on page 239.

3. Create a new record format as described in "Creating Record Formats" on page 262.

   The field names of the record format are used as the titles for the associated editable fields in the Edit Title Block dialog box.

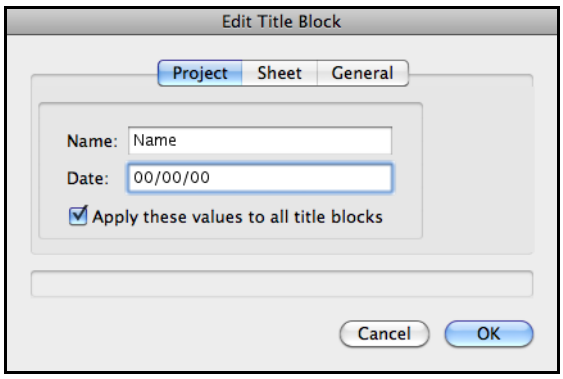


   To create a multi-line field, append a pound sign (#) to the field name, as in Drawing Title#.

   Use a P_ prefix to indicate project fields (these are fields with the same value on all title blocks in the file). An S_ prefix indicates sheet fields (these are fields with a different value on any title blocks). The prefixes cause the Edit Title Block dialog box, accessed from the Object Info palette, to be separated into project and sheet tabs. Fields without a prefix are placed on the General tab.

   Use an _SN suffix to indicate that a field should gets its value from the **Sheet Number** of its sheet layer. If **Use Automatic Drawing Coordination** is enabled in document preferences (Vectorworks Design Series required), the _SN suffix also means that when this title block field is updated, the **Sheet Number** of the sheet layer and of annotation objects on the layer are updated.

   Use an _SD suffix to indicate that the field should get its value from the **Sheet Title** of its sheet layer. If **Use Automatic Drawing Coordination** is enabled in document preferences (Vectorworks Design Series required), the _SD suffix also means that when this title block field is updated, the **Sheet Title** of the sheet layer is updated. If the field should also be multi-line, use an _SD# suffix.

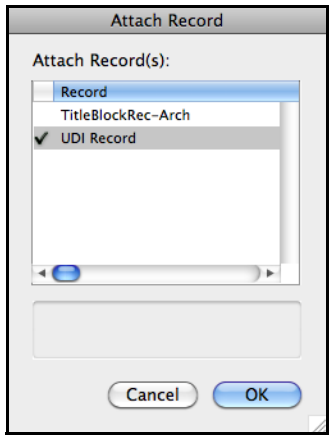


4. Creating a custom title block in the Vectorworks Design Series products requires additional steps, if the custom title block is to be used with the Issue Manager (see “The Issue Manager” on page 90), or contain issue fields or revision fields. When creating the record format to attach to the custom title block, fields linked to the Issue Manager require special prefixes or suffixes. If creating issue or revision fields, special text strings are added to the title block design. These record fields do not require linking, as they are automatically recognized by the Vectorworks program.

Title Block Area	Prefix Required
Issue Manager	If you are creating a US Arch-style title block with issue data linked to the Issue Manager: • Project fields (which contain the same value in all drawing set title blocks) must be prefaced with **P_** (example: P_Project Title) • Sheet fields (which contain different values in different drawing set title blocks) must be prefaced with **S_** (example: S_Sheet Scale) • Multi-line text fields (which contain multiple lines of text) must be appended with a # (pound sign) (example: P_Drawing Title#)
Issue Data	To add issue fields, such as issue number, date, note, and/or approvals to a sheet border which can be populated by the Issue Manager and edited manually, create text fields with special text strings to represent the issue fields and place these fields in the title block. The special text strings are (including colon): • Issue note **:iNote** • Issue number **:iNo** • Issue date **:iDate** • Issue approval **:iAppr** The text fields can have any justification, and should have word wrap on. The vertical alignment must be set to Top for the issue text to flow down, and to Bottom for the issue text to flow up. To enter or edit issue text, click **Edit Issue Data** from the Object Info palette of a selected sheet border (see “Editing Issue Data” on page 85), or use the Issue Manager. The issue history is a property of the sheet border, not the title block, so a title block is not required to enter issue data.

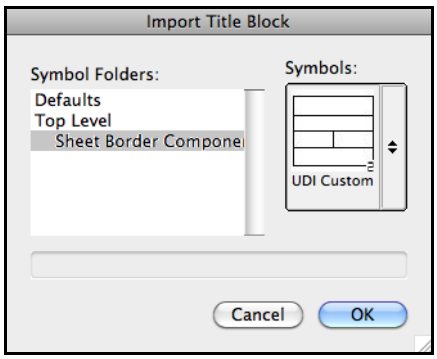
Title Block Area	Prefix Required
Revision Data	To add revision fields, such as revision number, date, zone, and/or approvals to a sheet border, create text fields with special text strings to represent the revisions and place these fields in the title block. The special text strings are (including colon): • Revision note **:rNote** • Revision number **:rNo** • Revision date **:rDate** • Revision zone **:rZone** • Revision approval **:rAppr** The text fields can have any justification, and should have word wrap on. The vertical alignment must be set to Top for the revision text to flow down, and to Bottom for the revision text to flow up. To add revision text to a sheet border, select **Use Revision Block** from the Object Info palette of a selected sheet border. To enter or edit the revision text, click **Edit Revision Data** (see "Editing Revision Block Data" on page 84). The revision history is a property of the sheet border, not the title block, so a title block is not required to enter revision data.

5. Attach the custom record format to the title block symbol through the Resource Browser as described in "Attaching Record Formats to a Single Symbol Instance or Object" on page 264.



6. Edit the title block symbol and attach the record fields to the associated symbol text; see "Linking Text to Record Formats" on page 269 for more information.
7. To be available to the **Sheet Border** tool, either the file must be saved in the sheet border- title block folder in the [Vectorworks]\Libraries\Defaults folder, or the custom symbol and its associated record format must be imported into the Custom Title Blocks.vwx file, located in the same default content folder.

   Alternatively, import the title block symbol from another file.
8. To place the custom title block, click **Title Block** from the Object Info palette of a selected sheet border, and select the custom title block symbol.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding a Title Block
Sheet Border Properties
The Issue Manager

The Issue Manager

The Issue Manager accesses and displays information about drawing issue, dates, and history on a sheet-by-sheet basis. It interacts with the sheet border object by allowing the user to control the sheet border title block data and issue information. (Sheet borders can either be placed with the **Sheet Border** tool in the Dims/Notes tool set—see "Adding a Sheet Border" on page 77—or created and placed by the **Document Setup** or **Create Standard Viewports** commands—see "Document Setup" on page 76 and "Standard Viewports" on page 186.)

To use the Issue Manager:

1. Ensure that at least one sheet border with a title block is present in the drawing.
2. Select **File > Issue Manager**.

   The Issue Manager dialog box opens.

   The tabs and fields displayed depend on the title block inserted in the sheet border. Custom title blocks may not match the parameters or tabs shown here. The fields and tabs shown assume that an Arch-style title block is inserted.
3. Click the Project Data tab to configure the project information.

   Leave a blank field to create an empty line in the title block.

Click to show/hide the parameters.

Parameter	Description
Project ID	Specifies the company project ID code
Project Title	Indicates the project title, which can include name and address information
Design Firm	Describes the name and address of the design firm
Project Manager	Provides the project manager's name
Designed By	Specifies the name of the lead designer
Total Sheets	Indicates the total number of sheet layers (or design layers named "sheet-") in the project
CAD File Name	Indicates the name of the file

The information on the Project tab applies to all sheet borders or title blocks in the set.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

4. Enter the project information. If the project was created with the **Create Standard Viewports** command (Vectorworks Architect required), some of the information has been automatically entered.
5. Click the Sheet Data tab to configure information about individual sheet layers in the set.

Click to show/hide the parameters.

Parameter	Description
Sheet Number	Specifies the sheet number
Sht Title	The drawing title can contain up to three lines
Sheet Scale	Displays the scale of the major drawing on this sheet
Consultant	Specifies the name and address of any consultants involved with this particular sheet
Drawn By	Provides the draftsperson's initials
Reviewed By	Specifies the reviewer's initials
Submitted By	Specifies the submitter's initials
Checked By	Specifies the checker's initials
Date	Specifies the date of the issue
Include in Current Issue	Select to include this sheet in the current issue set
< Prev / Next >	Click **Prev** and **Next** to switch to each sheet and enter the relevant data

6. Complete the information for each sheet in the drawing set to be issued. Toggle among other sheets in the drawing with the **Next** and **Prev** buttons.

   If the project was set up with the **Create Standard Viewports** command (Vectorworks Architect required), some of the information has been automatically entered. For each sheet, specify sheet-specific information and indicate whether to include the sheet in the issue.
7. Click the Issue Data tab to specify information about the current issue.

Click to show/hide the parameters.

Parameter	Description
Record the issue of this Drawing Set	Updates the title block issue data area
Issue Number	Specifies the number or letter of the current issue
Issue Date	Indicates the date of the current issue
Issue Note	Adds any comments or description of the issue
Record on checked sheets only	Updates only the issue data of sheets that were selected on the Sheet Data tab
Record on all sheets	Updates the issue data for all sheets in the file

8. Click **OK**.

   The sheet border title blocks are updated for the selected sheets. If specified, the issue information is included in the title block.

   Regardless of whether the title block contains fields for the issue data, the data is still written to a record attached to the sheet border, becoming a permanent part of the sheet border. If the title block changes later, the data can be displayed.

AL Standard Naming

The **Standard Naming** command controls the layer, class, and viewport names used in a project. These names can be changed to a user-defined system other than the default VWArch naming system. Standards can be mapped for office-wide use or to convert an existing file to the office standard. This command can also be used to assign specific attributes to classes in standards, and to change the names of layers, viewports/views and classes in the current standard. The command does not create new layers, classes, or viewports/views.

If classes, layers, and viewports/views have not been set up according to VWArch standards, their names may not match the example layer and class names presented here.

To set the standard naming of layers, classes, and views:

1. Select **File > Document Settings > Standard Naming**.

   The Standard Naming dialog box opens. Select a naming standard for the file, and choose whether auto-classing should be enabled for objects.

Click to show/hide the parameters.

Parameter	Description
Active Standard	Select the naming standard to apply to classes, design layers, sheet layers, viewports, and saved views
Details	Opens the Standard Naming Details dialog box; to edit the names for a custom standard (User 1, User 2, and User 3), the standard must be selected as the **Active Standard** first
Enable Auto-classing	Enables Auto-classing for the file and automatically places auto-classing objects into pre-assigned classes; see “Automatically Created Classes” on page 185. If objects were already present in the drawing when auto-classing is enabled, choose whether existing objects should be auto-classed when exiting this dialog box.

2. Click **Details** to edit custom class, layer, and viewport/view names.

   The Standard Naming Details dialog box opens, displaying class names on the **Classes** tab. Select the **Design Layers** tab to display design layer names, and the **Sheet Layers, VP, Saved Views** tab to display the names for viewports and their associated sheet layers, or for saved views.

   Using the reference list, verify the mapping of standard names. Custom naming can be specified for custom active standards by entering a new name for each layer, class, or viewport/view. If desired, class attributes can also be specified for each standard or custom class name. See “The Attributes Palette” on page 1093.

Click to show/hide the parameters.

Parameter	Description
Reference Standard	Select a reference naming standard from the list; the relevant class, layer, or viewport/view names are displayed below. Each active entry in the list is mapped to its corresponding reference list entry. User 1, User 2, and User 3 are custom standards. More custom standards can be defined (see “Creating Additional Custom Standards” on page 93).
Active Standard	Displays the overall naming standard currently in effect for the file (selected in the initial Standard Naming dialog box)
Reference Standard List	Lists the reference standard names for the class, design layer, viewport/sheet layer, or saved view

Parameter	Description
Active Standard List	Lists the standard name currently in effect and mapped to the reference standard for the class, design layer, or viewport/sheet layer, or saved view
Description of class/layer/view	Describes the currently selected class, layer, viewport/sheet layer, or saved view in the list
Edit Class/Layer/View Name	When a custom active standard is selected (User 1, User 2 or User 3), enter the custom name for the currently selected class, layer, or viewport/view
Class Attributes (Classes tab only)	Sets the attributes of a selected class in the **Active Standard** list
Lines	Select the line class attributes
Pen Color	Select the pen color
Line Weight	Select the line weight
Line Style	Select the line style
Fills	Select the fill style class attributes
Fill Pattern	Select the fill pattern
Fill Fore Color	Select the fill foreground color
Fill Back Color	Select the fill background color
Use at Creation	Applies the class attributes as the classed item is created

3. When the desired class attributes have been specified, and, for custom standards, the standard names have been established, click **OK**.
4. In the Standard Naming dialog box, click **OK**. If custom viewports exist in the drawing, you are prompted to run the **Create Standard Viewports** command again (Vectorworks Architect required; see “Creating Standard Viewports” on page 187) to update the viewports/views with the new naming standard, and update any new class attributes. If auto-classing was selected and objects had already been placed in the drawing, select whether to auto-class those existing objects.

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Additional Custom Standards
Coarse and Fine Custom Standards

A L Creating Additional Custom Standards

A custom naming standard can be created rather than using the VWArch or AIA/NCS standard. Although three layer, class, and viewport/view standards are available from within the Standard Naming dialog box (User 1, User 2 and User 3), up to ninety-nine can be created for layers, classes, and viewport/views by editing the ClassNameStds, LayerNameStds, and ViewNameStds worksheets. See “Using Worksheets” on page 1319.

To create a custom naming standard by worksheet:

1. In a new file, select **File > Document Settings > Standard Naming**.

   The Standard Naming dialog box opens.
2. Without making any changes, click **OK**.

   Three worksheets are created in the file, and are visible in the Resource Browser: ClassNameStds, LayerNameStds, and ViewNameStds.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Select one of the worksheets from the Resource Browser, and select **Resources > Open**.

   The worksheet opens for editing.

4. Highlight column **D (User 1)** and from the **Worksheet** menu select **Insert > Columns**. A new column is added in front of the selected one.

   New columns must be inserted after the **AIA/NCS** column and before the **Description** or **Pen Color** column.

5. Enter a name in cell **D1** for the new standard.

6. Enter a new standard name for each cell below **D1**.

   Cells left blank in the ClassNameStds worksheet will be assigned to the "None" class. Cells left blank in the LayerNameStds worksheet will be assigned to the "Layer-None" layer. Blank cells are not permitted in the ViewNameStds worksheet; if a worksheet with blank viewport/view names is attempted for use in Standard Naming, an error message is displayed.

7. The new standard is displayed in the Standard Naming dialog box.

   To use these changes in other files, either save the file as a template to be used as the basis for new drawings, or import each worksheet into the other file before running the **Standard Naming** command.

Coarse and Fine Custom Standards

When creating custom standard naming, naming standards may contain fewer standard names (coarser) or additional standard names (finer).

A "coarser" custom standard names two or more of the standard class, layer, or view names with the same user-defined name. Except for auto-classing, this is an irreversible process.

For example, the classes Area-Main, Area-Patterns, and Area-Spec can be combined into a single class called Areas. All objects assigned to the original three classes are reassigned to Areas.

When custom standard naming changes are complete, additional information displays to confirm any actions to be taken, such as merging changed class names to eliminate duplicates.

A "finer" standard maps a single class, layer, or view name to multiple names. Mapping is required to define the standard naming change, and the Standard Naming - Mapping dialog box opens automatically.

For example, an A-FP# layer AIA standard corresponds to two layer standards in the Vectorworks Architect product: Mod-Floor-# and Mod-Slab-#. If switching from the AIA standard to the VWArch standard, select the A-FP# layer from the list on the left and indicate the mapping for objects currently on the A-FP# layer by selecting the mapping layer on the right.

All objects assigned to the A-FP# layer are assigned to the Mod-Floor-# layer. Unmapped layers are not created.

AL Mapping Classes and Layers

The **Class and Layer Mapping** command "maps" any set of layers and/or classes in a drawing into a new layer and/or class. The file structure can be simplified, condensed, and/or renamed. Use this feature, for example, when a consultant is using DWG or DXF files. Complex, multi-classed files imported from DWG can be condensed or renamed, and empty classes deleted in a single operation. Mapping can be saved so the next time a file is received from that consultant, the file can be re-mapped in a single step. This same procedure can be revised for the process of providing files to consultants using a different file structure or drawing standard. See "Layers" on page 161 and "Classes" on page 176.

To modify layers and classes:

1. Select **Tools > Class and Layer Mapping**.

The Class Mapping dialog box opens. Modify the existing classes, and then click **Go To Layers** to open the Layer Mapping dialog box. The Layer Mapping dialog box is identical in layout to the Class Mapping dialog box.

Click to show/hide the parameters.

Parameter	Description
Existing Classes/ Layers	Lists the existing classes or layers in the file; layers or classes that are not moved to the Proposed or Merged list are deleted
Add >	Adds a layer or class from the list of existing layers or classes to the Proposed list
Merge >	Adds a layer or class to the Merged list; layers or classes in this list will be merged with the layer or class that is selected in the Proposed list
< Remove	Removes the selected item from the Proposed list
New	Creates a new class or layer in the Proposed list
Rename	Renames the selected proposed layer or class
Merged Items	Lists layers or classes to merge with the selected layer or class in the Proposed list. The merge occurs when the next action is performed in the dialog box, so that layers and classes are merged on an ongoing basis. Selecting a proposed layer or class lists its merged items again so that changes can be made, if desired.
Remove	Removes the selected item from the Merged list
Load	Opens a previously saved mapping file
Save	Saves the layer or class mapping as a file; layer and class mapping settings are saved separately. This saves time when making the same modifications to several drawings that are set up similarly.
Reset All	Moves all proposed and merged items back to the Existing list
Delete empty class/ layer in proposed drawing	Deletes all layers and classes that have no items associated with them
Go To Layers > /Go To Classes >	Toggles between the Class Mapping and Layer Mapping dialog boxes

When selecting existing and proposed classes and layers, press and hold the Shift key to select multiple, contiguous items or press and hold the Ctrl key (Windows) or Command key (Mac) to select non-contiguous items.

2. Click **OK**.

   The drawing is updated, using the new layer and class mapping. Layers and classes not moved to the Proposed or Merged list are deleted from the file, along with any objects in those layers or classes.

Spotlight Setup

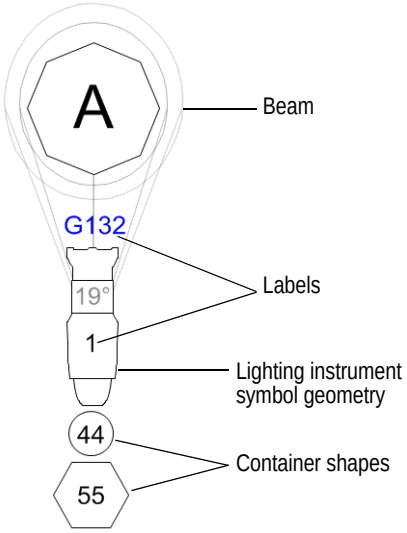
Lighting Device Setup

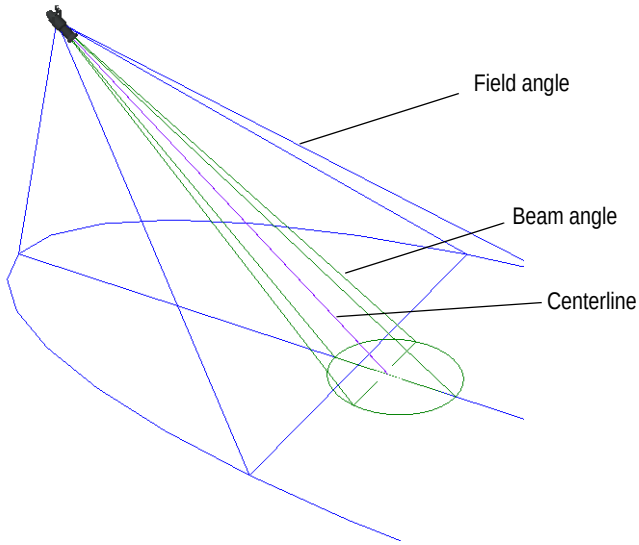
The options on the Lighting Device tab enable automatic positioning behavior, assign classes, and specify attributes of the lighting devices in the file. The parameters selected can have a powerful global effect on all the devices, both existing and new.

Lighting instruments can be automatically associated with a lighting position, both at insertion and when an existing instrument is moved on the light plot. At various times during the design work, particularly in the later stages, instruments may need to be moved slightly, but they should not be accidentally associated with a different nearby lighting position. The auto-positioning option provides the necessary control over the automatic association of instruments to lighting positions.

Class and color assignments for lighting instruments allow the designer to:

- Assign the lighting instruments to a specific class, or to one or more classes determined by an instrument parameter
- Control the color of lighting instruments, or parts of instruments, by gel color or from the Attributes palette
- Control the wireframe light beams, angles, and centerline of lighting instruments by class
- Control the label legend container appearance by class
- Control the appearance and visibility of lighting instruments in viewports, when using viewport overrides





The Object Info palette display of lighting device parameters can also be set from this tab.

To specify lighting device preferences:

1. Select **File > Document Settings > Spotlight Preferences.**

   The Spotlight Preferences dialog box opens. Click the Lighting Device tab.

Click to show/hide the parameters.

Parameter	Description
Enable lighting instrument auto positioning	Toggles the option to automatically associate lighting instruments with nearby lighting positions. When enabled, the instrument automatically associates with the closest lighting position within the **Pick radius**. When disabled, the instrument is assigned to a lighting position at insertion, but if moved later, the instrument's associated position remains unchanged. If instruments are still being added to a complex light plot, enable auto positioning, but decrease the radius value so that existing instruments can be moved without losing their pipe association. When instruments are only being repositioned, disable the auto-positioning option.

Parameter	Description
Pick radius	When auto positioning is enabled, specifies the radius for automatic association
Automatically assign the classes of all lighting instruments	Sets class options for lighting instruments; select the option to enable automatic classing options or deselect the option to turn off automatic class assignment
Use existing document class	Places all instruments in the selected class; select the class from the current list of classes or select New to create a new class
Use value of the field	Places all lighting instruments into a class that is determined by the value of an instrument parameter; select one of the instrument parameters from the list. For example, the value of the field **User Field 2** could determine the class of each lighting instrument; instruments whose **User Field 2** value is set to "FOH" will automatically be placed into a class together for automated appearance and visibility control. If the value of a particular field is not set (blank) for a lighting instrument, the instrument remains in the current class.
Class name	Specifies class naming options when the lighting instrument classes are determined by field value name; select a Prefix or Suffix and then enter the prefix or suffix name. Using a dash will nest the classes. For example, using a prefix of **Fixtures-** means that when a lighting instrument contains a value of Rental for **User Field 2**, the instrument is placed in a class named "Fixtures-Rental."
Modify lighting instrument color	Sets color options for the lighting instrument; select the option to enable automatic color options, or deselect the option to turn off automatic color control (the symbol specifies the instrument color)
Lighting instrument color set by: Lighting instrument	The Attributes palette controls the color of the lighting instrument geometry. Attributes can be set directly from the palette for selected lighting instrument or controlled by class (see "Setting Class Attributes" on page 181).
Lighting instrument color set by: Color field	The color specified in the lighting instrument's **Color** parameter also controls the appearance of the lighting instrument geometry
Use color field for the fill color	Applies the **Color** parameter value to the fill color of the lighting instrument
Use color field for the pen color	Applies the **Color** parameter value to the pen color of the lighting instrument
Modify only geometry in the class	Modifies the color of lighting instruments components within the selected class; select <All> to modify all the geometry, select the class from the current list of classes, or select New to create a new class
Label legend container	Controls the appearance of the label legend container(s) • Use symbol attributes: uses the container symbol definition to specify the attributes • Use label attributes: matches the attributes of the container to the attributes of the label legend text within the container • Use lighting instrument attributes: matches the attributes of the container to the attributes of the instrument
Show beam angle	Draws the beam angle for all lighting instruments with **Draw Beam** selected

Parameter	Description
Beam angle class	Controls the class of the wireframe representation of the beam angle; select a class for the beam, or create a new class. Alternatively, select <Lighting instrument Class>, which places the beam angle attributes in the same class as the instrument.
Show field angle	Draws the field angle for all lighting instruments with **Draw Beam** selected
Field angle class	Controls the class of the wireframe representation of the field angle; select a class for the beam, or create a new class. Alternatively, select <Lighting instrument Class>, which places the field angle attributes in the same class as the instrument.
Show centerline	Draws the centerline for all lighting instruments with **Draw Beam** selected
Centerline class	Controls the class of the wireframe representation of the centerline; select a class for the line, or create a new class. Alternatively, select <Lighting instrument Class>, which places the centerline attributes in the same class as the instrument.
Automatically Flip Front & Back Legend Text	Select to mirror the label legend along the X axis of all lighting instruments
Automatically Flip Left & Right Legend Text	Select to mirror the label legend along the Y axis of all lighting instruments
Additional Default Records	Opens the Additional Default Records dialog box, to select additional records with default data for lighting devices. Only applicable records display for selection. Select one or more record formats from those present in the file; a check mark in the **Use** column indicates that the default data in the record will be included. At insertion or replacement, the lighting device processes the additional record(s). If a match is found for a record field name, the default data from the record attached to the symbol applies to that field and displays in the Object Info palette.
Lighting Device Parameters	Opens the Lighting Device Parameters dialog box, to set lighting device parameter display options; see “Specifying Lighting Device Parameter Display” on page 98
Save as default	Saves the current Spotlight preferences as the default; the default settings saved include those on the Universe and Lightwright tab

2. Specify the auto-positioning and appearance settings for the file’s lighting devices.

   Click **Save as default** to use the lighting device and other Spotlight settings for other files created with the Vectorworks Spotlight product.

3. Click **OK**.

   Class setting changes require confirmation before regenerating the lighting devices.

Specifying Lighting Device Parameter Display

The lighting device parameters that display on the Shape tab of the Object Info palette can be customized. The customized parameters apply to all lighting devices in the file, and parameter sets can be saved.

To specify lighting device parameters:

1. Select **File > Document Settings > Spotlight Preferences.**
2. The Spotlight Preferences dialog box opens. Click the Lighting Device tab.
3. Click **Lighting Device Parameters**.

The Lighting Device Parameters dialog box opens. From this dialog box, Object Info palette parameters for lighting devices can be added, renamed, deleted, removed from display, and ordered. Parameter sets can be saved and managed.

Default lighting device parameters cannot be deleted.

Click to show/hide the parameters.

Parameter	Description
Settings list	Select a saved set of parameters to apply or edit. Once a saved set has been edited, <customized> displays in the list; click **Save** to save the custom set. Select <Default> to apply the standard set for the Vectorworks Spotlight program.
Save	Opens the Save Settings dialog box; enter the name for the new saved set. Enter the same name as a previously saved set to overwrite the set.
Manage	Opens the Saved Settings dialog box; select the set of saved parameters to rename or delete. Click **OK** to return to the Lighting Device Parameters dialog box.
Parameter list	Lists the parameters, in order, that will display on the Shape Tab of the Object Info palette. Only items with a check mark in the **Show in Shape Pane** column will display on the Object Info palette. To change the order of a parameter, click and drag within the # column.
New	Opens the Edit Field dialog box, to add a custom parameter to the list. Enter the name of the new parameter, and select its type and default value, if any (see "Creating Record Formats" on page 262). Click **OK** to return to the Lighting Device Parameters dialog box. To create a pop-up list of pre-defined parameter options, select the **Type** Pop-up and click **Choices** to open the Edit Choices dialog box. List the options and click **OK**.
Edit	Opens the Edit Field dialog box, to edit the currently selected parameter name, type, and default value.
Remove	Deletes the currently selected parameter(s) from the set

4. If settings have been changed, save the current settings as a set by clicking **Save**; <customized> is not considered as a saved set, though it applies to the current file.
5. Click **OK** to return to the Spotlight Preferences dialog box. The parameter display set that was selected applies to the current file.

   Regardless of whether **Save as default** is clicked in the Spotlight Preferences dialog box, the last selected saved set of parameters is in effect, even when creating a new file. New files created in Vectorworks Spotlight always default to the last saved parameter set.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Universe Assignment in Vectorworks Spotlight
Automated Import and Export to Lightwright
Lighting Instrument Properties
Record Formats

Universe Assignment in Vectorworks Spotlight

At the beginning of a project (and at any other time), the Vectorworks Spotlight product can be set to automatically handle the assignment of DMX universe and dimmer combinations. As instruments are added to the light plot, the **Universe** and **U Address** instrument parameters are determined based on the dimmer numbers specified in the Spotlight Preferences dialog box.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To specify the dimmer and universe assignment:

1. Select **File > Document Settings > Spotlight Preferences.**

   The Spotlight Preferences dialog box opens. Click the Universe tab.

Click to show/hide the parameters.

Parameter	Description
Manually assign the Dimmer and Universe	Disables automatic universe assignment by the Vectorworks Spotlight product; dimmer and universe values need to be assigned manually for each instrument
Automatically assign Universe	As addresses are entered for instruments, the universe is automatically assigned by the Vectorworks Spotlight product
System	Specifies the universe system; up to six systems, A through F, are available. Each system can contain up to 64 DMX universes.
Univ #, Label, Start #, End #	Specifies where the universe breaks occur. For each universe, enter the dimmer range with a **Start** and **End** value; divisions of 512 are typical, but non-uniform divisions are also accepted. The universe name (label) is not required.
< Previous Page /Next Page >	When defining more than eight universes, navigate to additional universe entries with **Previous Page** and **Next Page**. Up to 64 DMX universes can be specified per system.
Save as default	Saves the current Spotlight preferences as the default; the default settings saved include those on the Universe and Lightwright tab

2. To enable automatic universe assignment, click **Automatically assign Universe.**
3. Select a universe system and then for each universe in the system, provide a name (optional) and a starting and ending value. Click **Next Page** to continue entering universe break values.

   Click **Save as default** to use this universe assignment and other Spotlight preferences for other files created with the Vectorworks Spotlight product.
4. Click **OK**.

   The addresses of inserted instruments are automatically assigned to a universe. The **U Address** parameter in the Object Info palette displays the automatically assigned address when automatic assignment is enabled.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Device Setup
Automated Import and Export to Lightwright

Project Setup for Mechanical Design

At the beginning of a mechanical design project, use the **Drawing Setup** command to quickly set relevant drawing units, layer scale and print area from a single dialog box. A sheet border and title block can also be inserted automatically. An alternate method of setting up a document is to open one of the pre-configured machine design template files and use it as a starting point.

The Machine Design (Imperial) template opens with **File > Document Settings > Units** set to inches on the General Display tab, decimal rounding set to three decimal places, and the design layer scale set at 1:1.

To set up a project with the **Drawing Setup** command:

1. Select **File > New**.

   The Create Document dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Select **Create blank document**, and then click **OK**.
3. Select the **Drawing Setup** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Drawing Setup**
   - Landmark workspace: **Landmark > Machine Design > Drawing Setup**
   - Spotlight workspace: **Spotlight > Machine Design > Drawing Setup**

   The Drawing Setup dialog box opens.
4. Specify the drawing units, layer scale, print area, and sheet border, and then click **OK**. For more information on Drawing Units, Layer Scale, and the Print Area, see "Setting up the Drawing" on page 67.

Click to show/hide the parameters.

Parameter	Description
Drawing Units	The current drawing units value on the General Display tab of the Units dialog box displays; click **Change** to revise the current drawing units
Layer Scale	The current layer scale displays; click **Change** to set a different layer scale. If the active layer is a sheet layer, the layer scale cannot be changed.
Print Area	The current print area value displays; click **Change** to enter different print area parameters
Insert Sheet Border	Automatically inserts a sheet border
Insert Title Block	Select the title block to insert within the sheet border, or select None to leave out the title block; see "Adding a Sheet Border" on page 77
Insert Revision Block	Select to insert a revision history block within the sheet border; for more information, see "Adding a Sheet Border" on page 77
Insert Tolerance Block	Select to insert a tolerance specifications block within the sheet border (ASME title block required); for more information, see "Editing a Tolerance Block" on page 85
Insert Projection Block	Select to insert a projection block containing either a first angle or third angle projection symbol within the sheet border (ASME title block required)

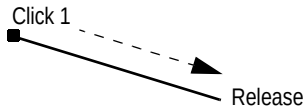
The drawing setup parameters are saved with the file. Use the Object Info palette to make any changes to the sheet border once it has been placed in the drawing.

Basic Techniques

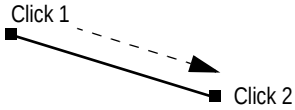
Using the Mouse

The easiest way to draw in the Vectorworks program is to use the mouse. Select the appropriate tool button and create an object in either the click-drag or the click-click mode, depending on the type of object.

In click-drag mode, click and continue to hold down the mouse button while you create an object; release the button when the object is the desired size and shape.



In click-click mode, click the mouse to mark the start point of an object, and then click again at each of the object's corners or vertices.



The program defaults to click-click mode, and all procedures in this guide are based on click-click drawing. This preference can be changed in Vectorworks preferences.

Use the Data bar for accurate object placement and creation. See "Using the Data Bar" on page 125.

Moving Around

There are several ways to move around within a drawing file, which allows you to look at the whole drawing or at select portions of it.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Panning

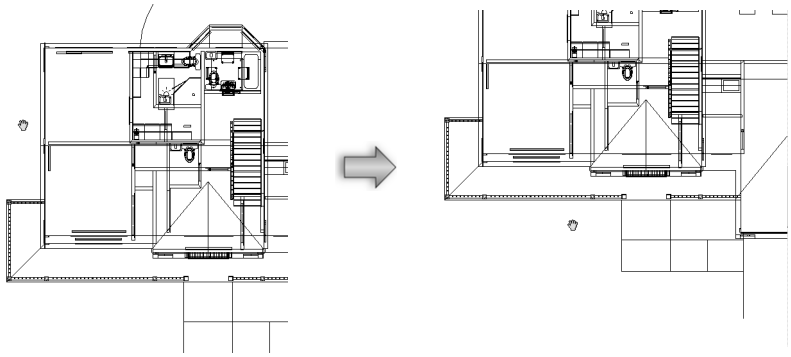
Use the **Pan** tool to move the drawing around the drawing window, changing the area of display. When in an uncropped perspective projection, the **Pan** tool moves the viewpoint in the model while the line of sight remains centered in the window; see "Projection" on page 1142.

 To pan around the drawing:

1. Click the **Pan** tool from the Basic palette.
2. Move the hand into the drawing window.
3. Click and hold down the mouse button, and drag the drawing around the screen.

   The rulers move with the page.
4. When the drawing shows the desired area, release the mouse button.

   Areas of the drawing that were off the page do not display until the mouse is released.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Double-click the **Pan** tool to refresh the drawing view.

Panning with the Mouse Wheel

On a wheel-mouse, click and hold down the mouse wheel to pan at any time, regardless of which tool is currently selected.

This feature will not work properly if the wheel button has been assigned a custom function in the mouse setup. For example, if the wheel button is set to perform a delete when clicked, a wheel click in the Vectorworks program deletes rather than pans. (The specific setting required for this feature depends on the type of mouse being used.)

Panning with the Arrow Keys

Use the arrow keys on the keyboard to pan at any time, regardless of which tool is currently selected. Specify the arrow shortcut keys for panning in Vectorworks preferences (see “Edit Preferences” on page 49). Press the shortcut key combination once to move the drawing one half-screen in the direction of the arrow.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Around

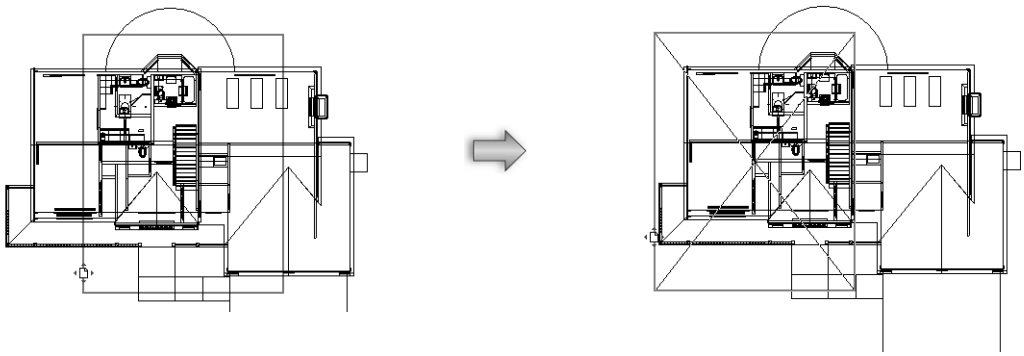
Moving the Page Print Boundary

The **Move Page** tool changes the position of the print boundary within the drawing area. Though the boundary moves, the rulers, user origin, and objects remain fixed. In 3D, the working plane does not move along with the boundary.

To move the page:

1. Click the **Move Page** tool from the Basic palette.
2. Click and drag the page outline to the desired location.

   The original position of the page continues to display, helping to measure your movement.
3. Click to set the position of the page.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Paste in Place works from user origin. If the page is moved, the image is placed in relation to the user origin, not the new page location.

Double-click the **Move Page** tool to set the page origin to be the same as the user origin.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Print Area
Moving Around

Zooming

Zoom controls the visual scale of a drawing. It does not affect the physical size of objects as set by the layer scale. Like a magnifying glass, zoom controls how close or far away objects appear on the screen. Zoom in to get a close-up view of a detail, and zoom out to get a broader view of the whole drawing.

During a zoom operation, the drawing continues to display normally if possible. A complex drawing may not display in full detail during a zoom operation, but as soon as the zoom is complete, the drawing displays normally. When in an uncropped perspective projection, zooming moves the viewpoint in the model while the line of sight remains centered in the window; see “Projection” on page 1142.

The Vectorworks program includes zoom functionality through the mouse wheel, through buttons on the View bar, and through the **Zoom** tool on the Basic palette.

Zooming with the Mouse Wheel

On a wheel-mouse, roll the mouse wheel forward to magnify the drawing (the **Zoom** tool on the Basic palette need not be selected). Roll the mouse wheel backward to zoom out from the drawing.

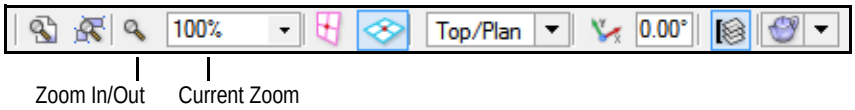
The mouse wheel behavior is controlled by the **Mouse wheel zooms** setting in the Edit tab of the Vectorworks Preferences dialog box. If the wheel is set to scroll by default, you must press the Ctrl (Windows) or Option (Mac) key while you roll in order to zoom. See “Edit Preferences” on page 49 for details.

> This feature will not work properly if standard scrolling is disabled in the mouse setup. For example, if the mouse’s scrolling size is set to “none,” mouse zooming in the Vectorworks program is disabled. (The specific settings required for this feature depend on the type of mouse being used.)

Zooming from the View Bar

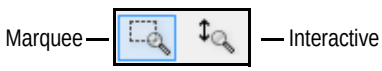
From the View bar, click the **Zoom** button to double the magnification of the drawing (a single-click on this button performs the same function as a double-click on the **Zoom** tool on the Basic palette). To reduce the magnification by half, press the Alt key (Windows) or Option key (Mac) when you click the **Zoom** button. To zoom by a specific amount, enter a zoom factor in the View bar, or select one from the pull-down list.

If an object or objects are currently selected, the zoom is centered on those object(s). To zoom on a specific area of the drawing (with nothing selected), click in an open area and then click the **Zoom** button.

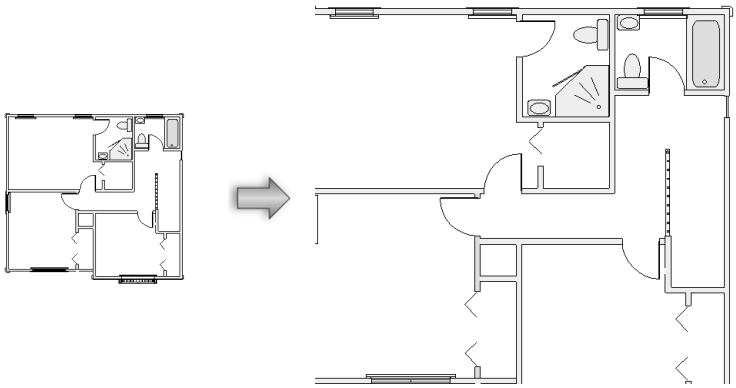


Zooming with the Zoom Tool

The **Zoom** tool has two modes.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mode	Description
Marquee	Magnifies the portion of the drawing that is within the marquee area; in click-drag mode, simply click once (do not create a marquee box) to double the zoom factor. To zoom out, hold down the Option (Mac) or Alt (Windows) key during the zoom.
Interactive	Interactively zooms the drawing area while moving the mouse



With the Zoom tool selected, double-click anywhere in the drawing to automatically activate the **Selection** tool.

Double-click the **Zoom** tool from the Basic palette to double the magnification of the drawing. The **Zoom In** button on the View bar performs the same action.

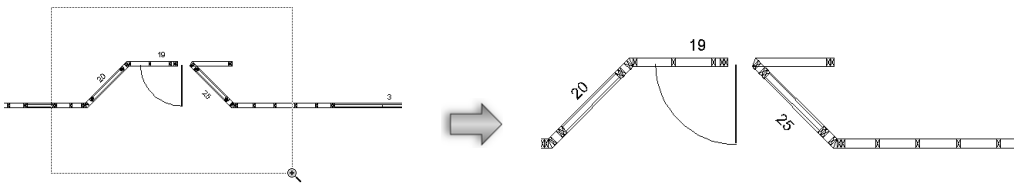
Marquee Mode

Select a portion of the drawing to magnify using marquee selection.

 To marquee zoom:

1. Click the **Zoom** tool from the Basic palette.
2. Click **Marquee** mode.
3. Click and drag to create a marquee box around the portion of the drawing to magnify.
4. Click again to zoom in.

   Vectorworks magnifies the selected section so that it fills the drawing window.



To zoom out, hold down the Option (Mac) or Alt (Windows) key while you draw a marquee box around a portion of the drawing. Click again to zoom out. The program reduces the magnification to display additional portions of the drawing around the selected section.

Interactive Mode

Choose an area of the drawing to magnify interactively.

 To interactively zoom:

1. Click the **Zoom** tool from the Basic palette.
2. Click **Interactive** mode.
3. Click in the drawing and hold down the mouse button (Mac) or left mouse button (Windows) while you move the mouse forward to zoom in on the drawing from the location of the click.

To zoom out, click in the drawing and hold down the mouse button (Mac) or left mouse button (Windows) while you move the mouse backward to zoom out of the drawing from the location of the click.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Around
Normal Scale

Normal Scale

The **Normal Scale** command automatically displays the drawing file at 100% of its real-world scale. For example, if a drawing scale is set to 1:1, every inch on the monitor corresponds to an inch on paper. Normal scale is the scale at which the drawing is printed.

To set the drawing to normal scale:

1. Select **View > Zoom > Normal Scale**.

   Vectorworks changes the view so that the objects' screen size and print size are the same.
2. If the drawing size is larger than the monitor's dimensions, scroll or pan around the drawing to see all elements.

Scrolling

Automatically Scrolling While Drawing

The Vectorworks program automatically scrolls the drawing area when an object is being drawn.

To use autoscroll:

1. Select any drawing tool.
2. Press and hold down the mouse button to begin drawing.
3. Drag the cursor on top of or past a scroll bar or ruler.

   The drawing window scrolls in the direction of the cursor.

   In click-click mode, if you do not hold down the mouse button while you draw, the autoscroll is halted if the cursor passes a scroll bar or ruler; this allows interaction with the scroll bars, tool palettes, and tool sets.

Using the Scroll Bars

Use the scroll bars to move the drawing around the screen. However, the scroll bars are best used for small movements, because they require the screen to redraw more frequently. For large movements, the **Pan** tool is faster. To display or hide the scroll bars, change the scroll bars setting in the Vectorworks Preferences dialog box.

Scrolling with the Mouse Wheel

On a wheel-mouse, press the Ctrl (Windows) or Cmd (Mac) key while you roll the wheel forward or backward to scroll the window up or down. To scroll the window left or right, press the Shift key while you roll.

The mouse wheel behavior is controlled by the **Mouse wheel zooms** setting in the Edit tab of the Vectorworks Preferences dialog box. The wheel can be set to scroll by default, so that no additional key must be pressed while you roll in order to scroll. See "Edit Preferences" on page 49 for details.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

This feature will not work properly if standard scrolling is disabled in the mouse setup. For example, if the mouse's scrolling size is set to "none," mouse scrolling in the Vectorworks program is disabled. (The specific settings required for this feature depend on the type of mouse being used.)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Around

Undoing and Redoing Actions

Specify preferences to control how many actions can be undone and redone.

Undoing Actions

One or more of the most recent actions can be undone. Specify the number of actions that can be undone on the Session tab of Vectorworks preferences. See "Session Preferences" on page 52 for information on this tab. The maximum number of actions that can be undone is 100. Select **Issue undo warnings** to open a warning dialog box when attempting to undo an action that cannot be undone.

To undo recent changes:

Select **Edit > Undo**.

The most recent change is undone. Continue to select the **Undo** command to undo changes in the reverse order in which they were performed.

The higher the maximum number of undos, the more memory may be required.

Redoing Actions

Actions that have been undone can then be redone. The number of actions that can be redone is determined by how many undos were performed.

To redo actions that were undone:

Select **Edit > Redo**.

The most recent undo is reversed and the action is executed again. Continuing to select the **Redo** command will redo changes in the reverse order that they were undone.

Selecting Objects

Use the **Selection** tool from the Basic palette to select objects for the next command or edit operation. The Shift key and the Alt (Windows) or Option (Mac) key are modifiers for selection actions. Create rectangular, lasso, or polygonal marquees around objects to select single or multiple objects. The following table describes the various selection methods.

Method	Selection Action
Click	Standard selection method; selects a single object only
Option-click (Mac) or Ctrl-click (Windows)	Creates a copy of the object and places it directly over the selected object, unless the preference has been disabled (see "Edit Preferences" on page 49)
Shift-click	Selects multiple objects as each object is clicked; also can be used to deselect one or more objects without affecting other selected objects
Option-drag (Mac) or Ctrl-drag (Windows)	Places a copy of the object where the mouse button is released
Rectangle, lasso, polygon marquee	Selects all objects that are completely contained within the marquee
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Method	Selection Action
Shift-marquee	Reverses the selection status of objects inside a marquee; if objects inside the marquee are selected, this method deselects those objects
Option-marquee (Mac) or Alt-marquee (Windows)	Selects all objects that the marquee passes through, as well as those contained within the marquee
Invert Selection command (on the **Edit** menu)	Deselects everything that is currently selected, and selects all visible objects in editable layers and classes that are not currently selected
Double-click	Reactivates the working plane on which a planar object was created; may also have other behavior such as symbol editing or polygon reshaping, depending on selected settings
Select Vertex in Object Info Palette (context menu command)	For vertex-based objects, selects a single vertex for editing on the Object Info palette; see “Editing Vertex-Based Objects” on page 1002
Force Select (context menu command)	Activates the object’s class, layer, or both (as necessary) and selects the object, even if the object could not normally be selected due to a different layer scale, or the current class or layer option settings

To deselect all selected objects, double-click on the **Selection** tool, click in an empty area of the drawing, or press the “x” key twice in rapid succession.

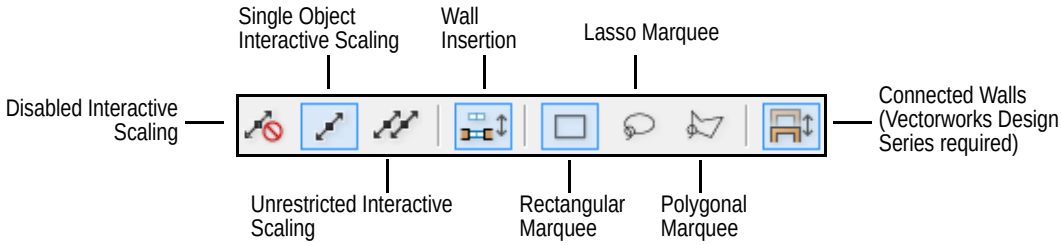
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

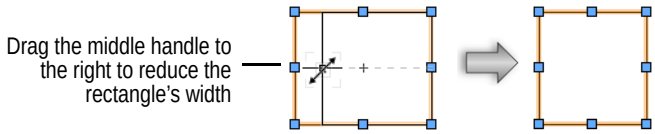
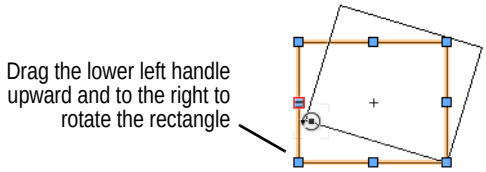
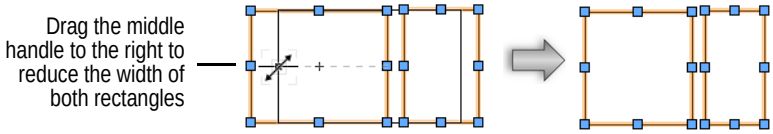
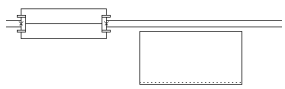
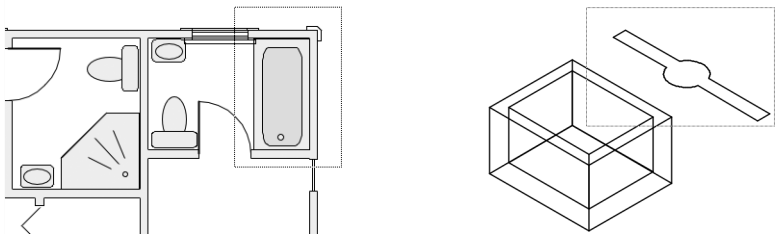
The Selection Tool

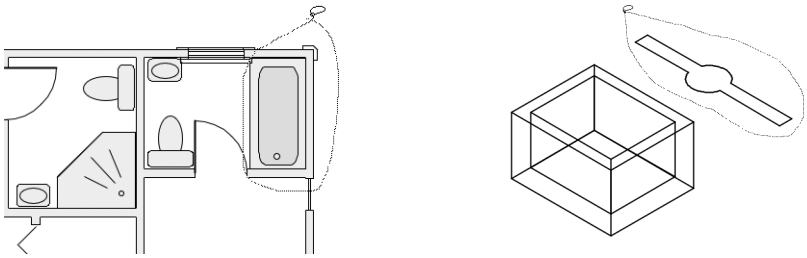
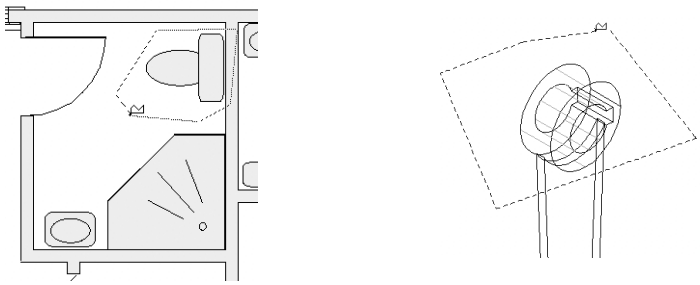
Use the **Selection** tool to select objects, to move and resize objects in any view, and to insert objects in or next to a wall.

In a rendered 3D view, hidden geometry located behind solid objects cannot be directly selected. This makes it easier to select solid objects without the interference of hidden objects. To select hidden geometry, use X-ray Select mode (see “X-ray Select Mode” on page 113) or the clip cube feature (see “Viewing a Model with the Clip Cube” on page 1155).

For marquee selection, press the Option (Mac) or Alt (Windows) key while drawing the marquee to select all objects that intersect the marquee.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mode	Description
Disabled Interactive Scaling	No reshape handles display for a selected object, so that it can be dragged from any point without accidentally affecting its size
Single Object Interactive Scaling	Allows resizing by dragging a reshape handle that displays on a single selected object.  Allows symbols to be scaled, similar to the **Scaling** option in the Object Info palette, unless the **Allow Interactive 2D symbol scaling** Vectorworks preference has been disabled; see "Edit Preferences" on page 49. Page-based symbols can only be scaled symmetrically. To rotate certain objects (such as rectangles, rounded rectangles, ovals, text objects, bitmaps, and PICT images), press the Option key (Mac) or Alt key (Windows) while dragging a reshape handle. 
Unrestricted Interactive Scaling	Allows resizing of multiple objects by dragging a reshape handle that displays on one of the selected object(s) 
Wall Insertion	Allows symbols and plug-in objects that are already on drawing to be placed into a wall segment. When disabled, symbols can be moved next to or on a wall without becoming part of the wall. See "Wall Insertion Mode" on page 246 for more information. 
Rectangular Marquee	Creates a marquee box around objects when selecting. Click to set the start point, drag the mouse in the desired direction, and release to set the end point. All objects within the marquee are selected. 

Mode	Description
Lasso Marquee	Creates a free-form marquee, allowing a more exact selection of irregular shapes. Click to set the start point, drag the mouse in the desired direction, and release to set the end point. All objects within the marquee are selected. 
Polygonal Marquee	Creates a marquee with an irregular polygonal shape. Click to set the start point, and then continue clicking to define the shape. Double-click to finish the marquee. All objects within the marquee are selected. 
Connected Walls (Vectorworks Design Series required)	Connected Walls mode becomes available when the Vectorworks Architect or Landmark product is installed. See “Moving Connected Walls” on page 541 for information.

To select objects:

1. Click the **Selection** tool from the Basic palette.
2. Select the appropriate mode.
3. Select the desired object(s).

   Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selecting Objects
Select All
Previous Selection
Coincident Object Selection
X-ray Select Mode
Selection and Pre-selection Indicators

Select All

The **Select All** command selects all visible objects in editable layers and classes. The active layer is always editable, but other layers can be editable if they are visible and if the **Layer Options** are set to Show/Snap/Modify Others. For more information about setting layer and class visibility and layer and class options, see “Setting Visibilities in the Organization Dialog Box” on page 193, “The Visibility Tool” on page 195, and “Setting Class and Design Layer Options” on page 193.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To select all objects in the drawing area:

Select **Edit > Select All**.

Alternatively, right-click (Windows) or Ctrl-click (Mac) on the drawing area, and select **Select All** from the document context menu.

The program selects all visible objects that can be modified. Each of these selected objects is displayed with highlighting. Locked objects are also selected so they can be unlocked for modification.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selection and Pre-selection Indicators
Selecting Objects
The Selection Tool

Previous Selection

The **Previous Selection** command reselects the set of objects that were most recently selected. This is especially useful if a large number of objects were accidentally deselected.

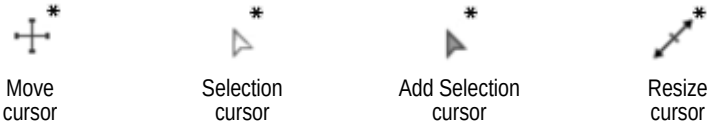
To reselect previously selected objects, select **Edit > Previous Selection**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selection and Pre-selection Indicators
Configuring Interactive Display
Selecting Objects
The Selection Tool

Coincident Object Selection

When the edges or points of multiple objects exist at the same location on the drawing, you might not be able to simply click with the **Selection** tool to select the object(s) you need. The desired object might have an edge that coincides with the edge of another object, for example. In this situation, an asterisk (*) is added to the various selection cursors to indicate that multiple objects exist at the current location, as follows.



You can then use either the Coincident Selection key or the **Select Coincident Objects** command from the context menu to open a dialog box from which you can select the appropriate objects. Coincident selection can work in conjunction with X-ray Select mode when coincident objects are hidden by rendered solids (see “X-ray Select Mode” on page 113).

To select one or more coincident objects:

1. Click the **Selection** tool from the Basic palette.
2. When the cursor indicates that multiple objects are available beneath it, press and hold the Coincident Selection key (J by default) and click the drawing. Alternatively, right-click (Windows) or Ctrl-click (Mac) and select **Select Coincident Objects** from the object context menu.

   The Coincident Selection key can be changed in the Workspace Editor; see “Modifying Snapping and Mode Shortcuts” on page 1842.
3. The Select dialog box opens. Each object available for selection at that point displays in a list, in top to bottom stacking order.
4. Click to select one or more items from the list; the selected objects are highlighted in the drawing.
5. Click **OK** to close the dialog box and select the objects.

Alternatively, double-click an item from the list to select it in the drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

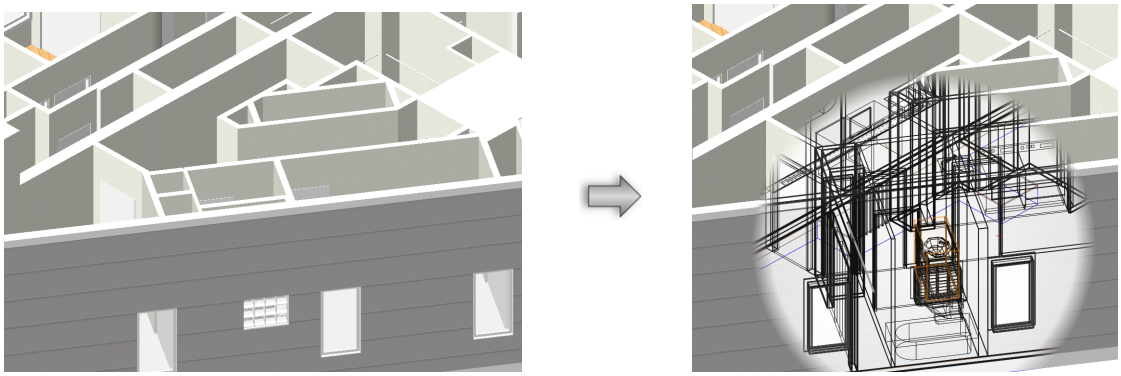
Selection and Pre-selection Indicators
Selecting Objects
The Selection Tool
X-ray Select Mode

X-ray Select Mode

This feature is available on all Mac systems, and on Windows systems only when the GDI+ imaging preference is enabled. (See “Vectorworks Display Preferences” on page 50 for information about setting this preference.)

A 2D object in Top/Plan view can have a fill that obscures objects beneath it on the same design layer or on other visible layers. In a 3D view, rendered objects hide or “occlude” objects behind them. To make it easier to see these hidden objects and select them with the **Selection** tool, use the X-ray Select mode. When the designated key is pressed in Top/Plan view, all 2D objects with fill become semi-transparent. In a 3D rendered view, pressing the key creates an area around the cursor where you can view the objects behind the solid object(s) in Wireframe mode, as if you had X-ray vision.

This is a convenient way of selecting objects without switching to Wireframe render mode.



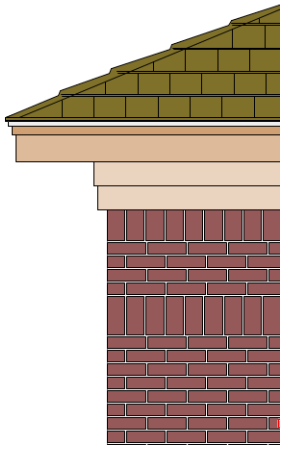
In addition to the **Selection** tool, the X-ray Select mode can be used with other tools that require the selection of objects, such as the **Eyedropper**, **Visibility**, and **Select Similar** tools.

To see and select objects with the X-ray Select mode:

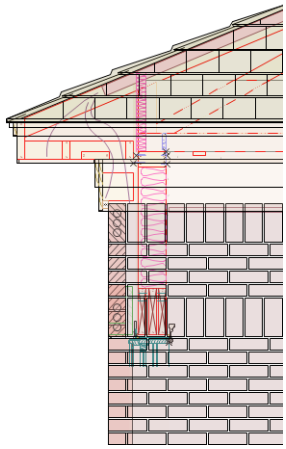
1. Click the **Selection** tool from the Basic palette.
2. Press and hold the X-ray Select mode key (the B key, by default). In Top/Plan view, all 2D objects that have a fill become semi-transparent (edge lines are unaffected); in a 3D view, rendered objects display in Wireframe mode near the cursor.

   The X-ray Select mode key can be changed in the Workspace Editor; see “Modifying Snapping and Mode Shortcuts” on page 1842.

   The opacity of fills in X-ray Select mode can be changed in the Interactive Appearance Settings dialog box; see “Configuring Interactive Display” on page 116.
3. Click or Shift-click to select objects as needed. Objects are selected the same way they are normally, except that a filled object that is beneath another object behaves as if it has no fill; you must click the edge of the object to select it.
4. Release the key to exit X-ray Select mode.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Layer 1 has an exterior view of a building made from filled objects. Even though the layer options are set to **Show/Snap/Modify Others**, the objects on Layer 2 are not visible and cannot be selected by clicking with the **Selection** tool.



Layer 2 has a section view of the same building, also made from filled objects. When the X-ray Select mode key is pressed, all filled objects become transparent. The objects on Layer 2 are visible and can be selected by clicking with the **Selection** tool.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selection and Pre-selection Indicators

When you move the cursor over the objects in a drawing, and when you select objects, highlighting and handles provide information about the current state of the objects. These indicators can be customized as needed in the Vectorworks preferences. See “Interactive Preferences” on page 56 for details.

Highlighting

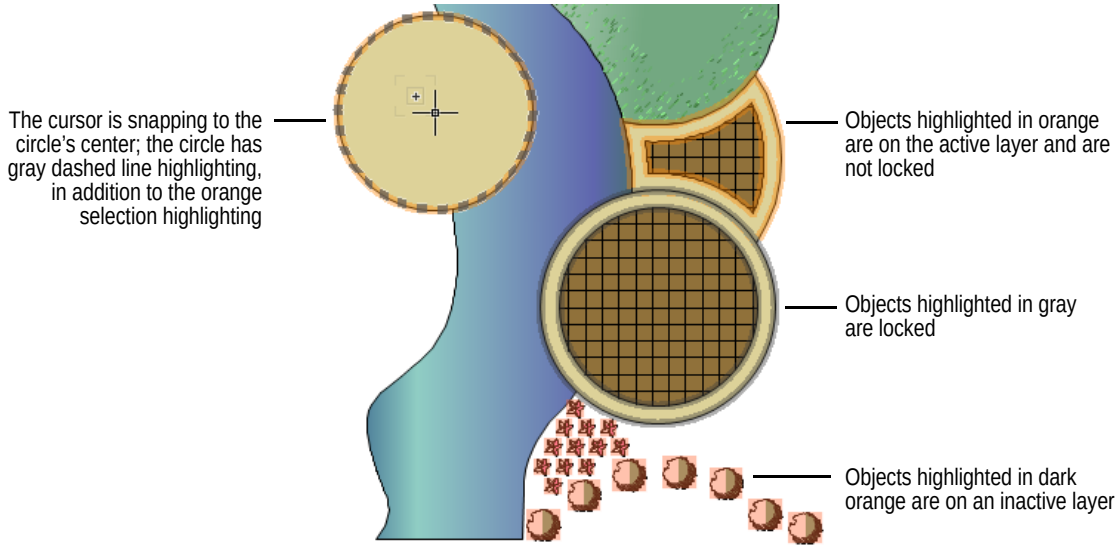
Highlighting can indicate the following information about an object in a drawing.

- Currently selected
- Currently “pre-selected” because it is under the cursor (or within the selection marquee)
- Locked
- On an inactive layer
- Causing one of the displayed “snap points”
- Can be acted upon by the currently active tool

In the Vectorworks preferences, there are several options to customize the highlighting.

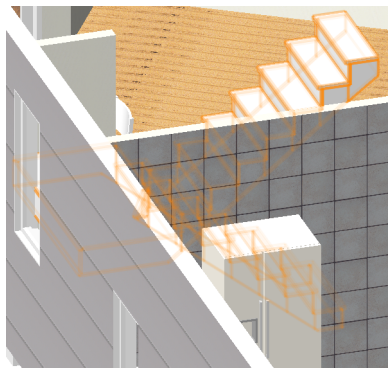
- The different types of highlighting can be enabled and disabled individually.
- The highlighting for selected objects can be animated.
- The timing of the highlighting for pre-selected objects can be adjusted.
- The color, opacity, and size of each type of highlighting can be adjusted.

By default, all highlighting options are enabled, and selected objects display as shown in the following illustration.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Sample highlighting with several types of objects selected

In a rendered 3D view, solid objects "occlude" or hide objects behind them. Highlighting of occluded objects is set, by default, to be less opaque than that of objects not occluded behind other objects.



The selected stair is partially occluded by the walls and other objects; selection highlighting is not as opaque in those areas

Handles

In addition to highlighting, square "handles" can display on selected objects. The appearance of the handles is controlled by the **Selection highlighting** setting on the Interactive tab in the Vectorworks preferences. Also, the colors of the handles can be customized.

If selection highlighting is enabled (the default setting):

- Handles display on a selected object if the object can be edited with the active tool
- Unlocked objects on the active layer have blue handles
- Unlocked objects on an inactive layer have white handles
- Locked objects have no handles (since they cannot be edited)
- Handles can be used as reshape points

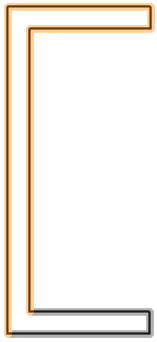
If selection highlighting is disabled:

- Handles display on all selected objects
- Unlocked objects on the active layer have blue handles
- Locked objects on the active layer have gray handles
- All objects on an inactive layer have white handles

- Some of the handles can be used as reshape points, and some cannot, depending on the object, and on which tool is active

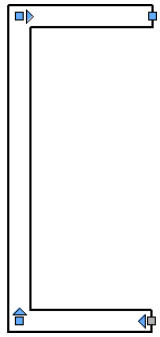
Selection highlighting enabled

The selected wall is highlighted in orange; blue handles indicate that the wall can be reshaped with the active tool



With three walls selected, the locked wall (bottom) is highlighted in gray; there are no handles because the objects cannot be reshaped with the active tool

Selection highlighting disabled



With three walls selected, the locked wall (bottom) has a gray handle on the right side; the unlocked walls have blue handles

Configuring Interactive Display
Selecting Objects

Configuring Interactive Display

You may find it helpful to customize the appearance of interactive elements in the program, including general elements, object highlighting, SmartCursor elements, and snap points.

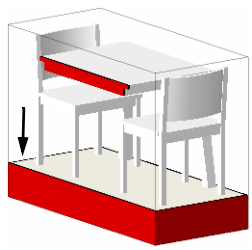
To configure the appearance of interactive elements:

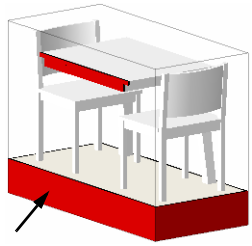
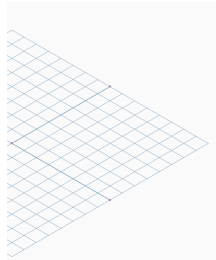
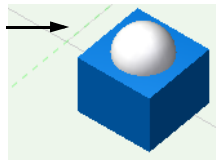
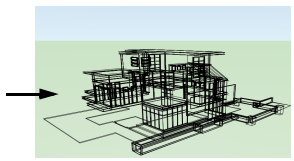
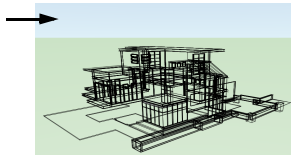
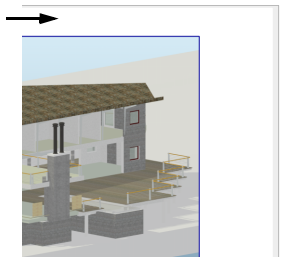
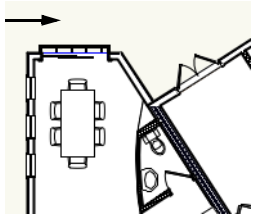
1. Select **Tools > Options > Vectorworks Preferences**. On the Interactive tab of the Vectorworks Preferences dialog box, click **Interactive Appearance Settings**.

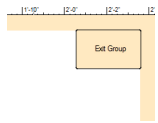
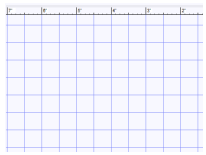
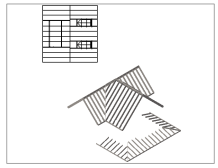
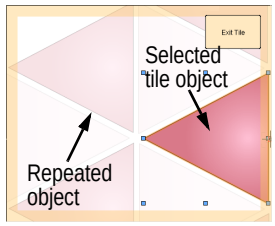
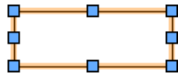
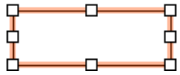
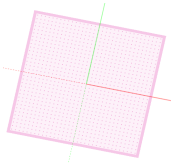
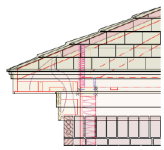
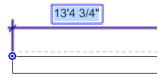
   Alternatively, select **Tools > SmartCursor Settings**. On the General tab of the SmartCursor Settings dialog box, click **Interactive Appearance Settings**.

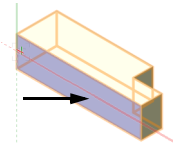
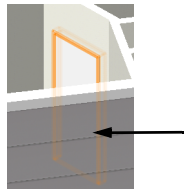
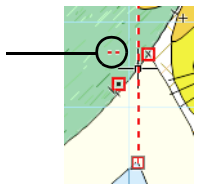
   The Interactive Appearance Settings dialog box opens.

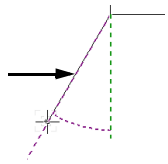
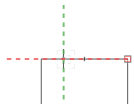
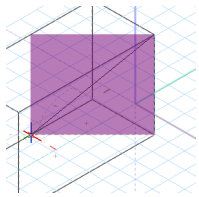
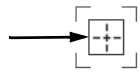
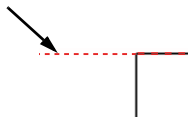
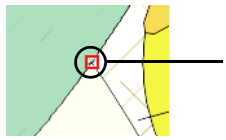
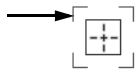
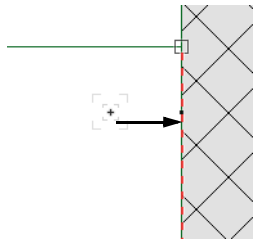
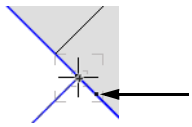
2. Select either the Standard Background or Black Background tab, depending on whether the black background display preference is enabled (see “Vectorworks Display Preferences” on page 50). The same elements can be customized for both background types, except that there are no options for setting background colors for black backgrounds.

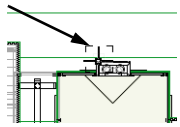
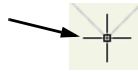
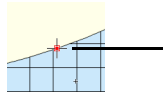
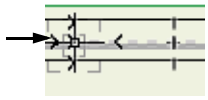
Element	Description	Example
Clip Cube		
Edges	Sets the color for edges where surfaces intersect a face of a clip cube	

Element	Description	Example
Solid Fill	Sets the color for the area where solid objects intersect a face of a clip cube	
General		
Active Layer Plane	Sets the color and opacity of the active layer plane grid	
Background - 3D Views (Standard Background only)	Sets the background color of the drawing area for design layers in 3D views	
Background - Perspective Horizon Ground	Sets the ground color and ground horizon color when in non-cropped perspective view; the foreground color gradually transitions to the background color as a gradient in the direction of the horizon	
Background - Perspective Horizon Sky	Sets the sky color and sky horizon color when in non-cropped perspective view; the foreground color gradually transitions to the background color as a gradient in the direction of the horizon	
Background - Sheet Layers (Standard Background only)	Sets the background color of sheet layers	
Background - Top/Plan (Standard Background only)	Sets the background color of the drawing area for design layers in Top/Plan view	

Element	Description	Example
Editing Modes Border	Sets the color, opacity, and thickness of the colored border when in an editing mode	
Grid	Sets the color and opacity of the reference grid	
Page Boundary	Sets the color and opacity of the page boundary border	
Repetition Opacity	Sets the opacity for the repetitions of the selected line type or tile object, which are displayed when a line type or tile resource is being edited	
Reshape Handles - Active Layer	Sets the color of the square handles that can be dragged to reshape a selected object on the active layer	
Reshape Handles - Inactive Layer	Sets the color of the square handles that can be dragged to reshape a selected object on an inactive layer	
Working Plane	Sets the color of the working plane and opacity of its fill and frame, as well as the opacity and thickness of the working plane axes	
X-ray Select Mode	Sets the opacity of the objects in X-ray Select mode	
Object Highlighting		
Action	Sets the color, opacity, and thickness of the highlighting for objects being modified	
Active Layer	Sets the pattern, forecolor, backcolor, opacity, and thickness of the highlighting for an object that is selected on the active layer	
Inactive Layer	Sets the pattern, forecolor, backcolor, opacity, and thickness of the highlighting for an object that is selected on an inactive layer	

Element	Description	Example
Automatic Plane Source	Sets the color and opacity of the automatic working plane	
Locked	Sets the pattern, forecolor, backcolor, opacity, and thickness of the highlighting for an object that is locked	
Pre-Selection - Active Layer	Sets the color, opacity, and thickness of the highlighting for an object that is pre-selected on the active layer; an object can be pre-selected by being either under the cursor's selection box, or inside a selection marquee	
Pre-Selection - Inactive Layer	Sets the color, opacity, and thickness of the highlighting for an object that is pre-selected on the inactive layer	
Pre-Selection - Locked	Sets the color, opacity, and thickness of the highlighting for an object that is locked	
Snap Object	Sets the color, opacity, and thickness of the highlighting for an object that is being snapped to; the highlighting is always a dashed line	
Tool Highlighting	Sets the color, opacity, and thickness of the highlighting for an object that can be edited by the active tool	
Occluded 3D Graphics	Sets the opacity of feedback graphics, such as highlighting, snapping, selection indication, and some preview graphics, when occluded by a rendered 3D solid	
SmartCursor		
Acquisition Hints	Sets the color, opacity, and thickness of the graphical hints which display near the cursor to show when a smart point, smart edge, or vector lock can be acquired or released	
Cue Background	Specifies the background color and opacity of SmartCursor cues	
Cue Text	Sets the color and size of SmartCursor text	
Datum Indicator	Sets the color, opacity and thickness of the datum indicator	

Element	Description	Example
Extension Lines - Angle	Sets the color, opacity and thickness of angle extension lines	
Extension Lines - Smart Point	Sets the color, opacity and thickness of smart point alignment lines (the selected color may be overridden by the default X, Y, and Z axis colors)	
Planes	Sets the color and opacity of angle and alignment planes (the selected color may be overridden by the default X, Y, and Z axis colors)	
Selection Box	Sets the color, opacity, and thickness of the selection box around the SmartCursor	
Smart Edges	Sets the color, opacity and thickness of smart edge lines	
Smart Points	Sets the color, opacity, and thickness of smart points	
Snap Box	Sets the color, opacity, and thickness of the snap box around the SmartCursor	
Vector Locks	Sets the color, opacity, and thickness of vector locks	
Snap Points		
Available	Sets the color, opacity, and thickness of snap points available near the cursor and within the snap box	

Element	Description	Example
Congestion Indicator	Sets the color, opacity, and thickness of the snap box when there are too many available snap points around the SmartCursor (using the snap loupe is recommended in this situation; see “Using the Snap Loupe” on page 148)	
Current	Sets the color, opacity, and thickness of the current snap indicator	
Obtained	Sets the color, opacity, and thickness of the acquired snap point	
Special	Sets the color, opacity, and thickness of snap point hints such as midpoint, center, along line, and tangent snaps	
Reset	Restores the default settings for all interface elements	

3. Click the column heading to sort by interface element, if desired. Select the element to change, and then set the parameters that display to the right of the element list as needed. Different parameters display, depending on the element selected.

Click to show/hide the parameters.

Parameter	Description
Pattern / ForeColor / BackColor	To use a solid color, select the color from the **ForeColor** menu; then from the **Pattern** menu, select the solid pattern for that color To use a pattern, select the **ForeColor** and **BackColor** colors from the menus (see “Selecting a Color from a Color Palette” on page 1135); then select a **Pattern**
Color	Select the color (see “Selecting a Color from a Color Palette” on page 1135)
Opacity	Drag the slider right to increase or left to decrease the opacity of the interactive element
Size	Drag the slider right to increase or left to decrease the width of the interactive element

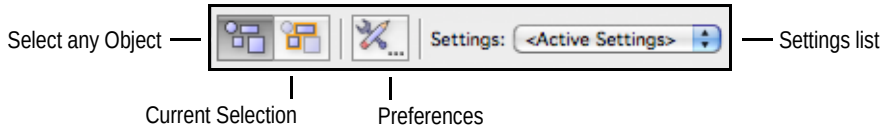
4. Click **OK** to save the settings, which apply to all Vectorworks drawings.
5. From the Vectorworks Preferences dialog box, click **OK** again to save the preferences.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selection and Pre-selection Indicators
Selecting Objects
Setting Snapping Parameters
Drawing with Snapping
The Selection Tool
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selecting Similar Objects

Use the **Select Similar** tool to select objects that have features in common with a specific object. This can be useful if you need to move or modify a large number of similar objects, such as windows. It can also make it easier to locate similar objects in a drawing that is very crowded.



Mode	Description
Select any Object	Selects objects that are similar to the clicked object
Current Selection	Selects objects that are similar to the clicked object, within a group of preselected objects
Preferences	Sets the default parameters to be used for the tool
Settings	Sets the group of parameters to be used for the tool—either the currently active settings, or a selection from the list of saved settings

Walls and objects within walls cannot be selected at the same time. Therefore, sometimes the objects selected by this tool may not be as expected. For example, you cannot select a set of walls and then use Current Selection mode to select all windows within those walls.

To select similar objects:

1. To select from a specific set of objects, select the objects first.
2. Click the **Select Similar** tool from the Basic tool palette.
3. Click **Preferences** from the Tool bar to specify the set of object attributes to match. The Select Similar Preferences dialog box opens.

   Alternatively, select a set of saved attributes from the **Settings** list on the Tool bar.

Click to show/hide the parameters.

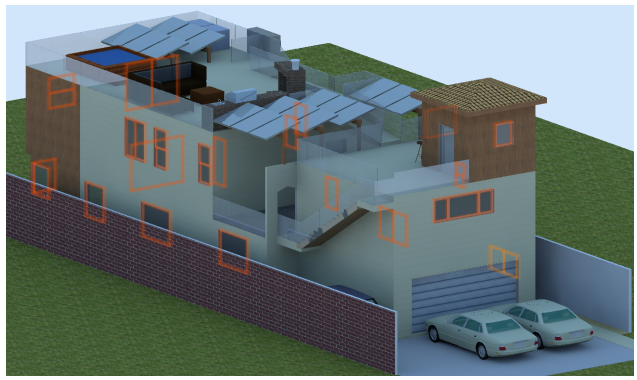
Parameter	Description
Saved Settings Options	
Settings	Select <Active Settings> to use the attributes currently selected in the dialog box; to use a set of saved attributes, select them from the pull-down list
Save	Opens a dialog box to name and save the currently selected attributes so that they can quickly be selected as a set
Manage	Opens the Saved Settings dialog box to rename or delete sets of saved tool attributes
All	• **Checked box:** All attributes in the group are selected; click to deselect all attributes in the group. • **Empty box:** No attributes in the group are selected; click to select all attributes in the group. • **Box displays (Windows) or (Mac):** One or more attributes in the group are not selected; click to select all attributes in the group.

Parameter	Description
Fill Attributes	Selects objects with matching fills, including foreground and background colors, styles (pattern, hatch, gradient, or image fill), and textures
Text Attributes	Selects text objects, dimensions, or plug-in objects with matching text attributes, including font, size, font style, alignment, and spacing
Wall Attributes	Selects walls and round walls with matching thicknesses and fill styles
Pen Attributes	Selects objects with matching pen foreground and background colors, styles (solid, pattern, or line type), and line weights
Other	Selects objects with various other matching attributes
Line End Marker	Selects line objects with matching end markers, including marker styles and sizes

4. Select the similar attributes to use for the selection, and click **OK**.
5. Click the object that is similar to the objects you want to find.

   An alert displays if the clicked object is not appropriate for the selected attributes (for example, if the **Font** attribute is selected, but a plant object is clicked).
6. The similar objects are selected.

With the **Select Similar** tool set to select by similar object type, click on a window



All window objects are selected (including those behind solid objects)

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Managing Saved Settings

Managing Saved Settings

If you have saved attributes to use with the **Select Similar** tool, use the **Manage** button on the Select Similar Preferences dialog box to rename or delete these saved settings when needed.

To manage saved settings for the **Select Similar** tool:

1. Select the **Select Similar** tool from the Basic tool palette, and then select **Preferences** on the Tool bar. The Select Similar Preferences dialog box opens.
2. Click **Manage**.

   The Saved Settings dialog box opens. Select the set of saved attributes to change.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Rename	Select and enter a new name for the set of attributes. If the entered name is already assigned to another set of attributes, you are prompted to confirm that you want to replace the existing set with the set being saved.
Delete	Select to delete the set of attributes. You are prompted to confirm that you want to delete the saved settings.

3. Click **OK** to close the Saved Settings dialog box. Click **OK** again to close the Select Similar Preferences dialog box.

Selecting Similar Objects

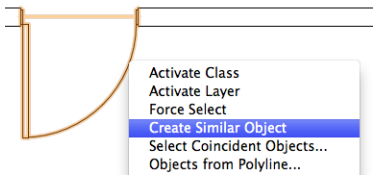
Creating Similar Objects

The **Create Similar Object** command is an easy way of creating an object that matches an object that is already in a drawing. The associated tool, tool mode, and object attributes and parameters are automatically set to create an object that matches the source object.

To create an object similar to an existing object:

1. With the cursor over the source object, either right-click (Windows) or Ctrl-click (Mac) and select **Create Similar Object** from the context menu. The source object can be visible or grayed, and does not need to be in the current layer.

   Alternatively, press the Ctrl + Alt (Windows) or Cmd + Option (Mac) shortcut key combination while clicking over the source object.



2. The tool required to create the object is automatically activated.

   Any tool settings, as well as any global settings used by the tool, change to match the source object, as shown here.

Source Object or Attribute	Result
Graphical attributes	Updates the global Attributes palette settings to match those of the source object, including by-class or class settings
Class	Sets the active class to that of the source object
Layer	Sets the active layer to that of the source object, if the Vectorworks preference **Change active layer for Similar Object Creation** is enabled; see “Session Preferences” on page 52
Text, or object with text set by global font settings	Matches the global font settings of the source text or object text

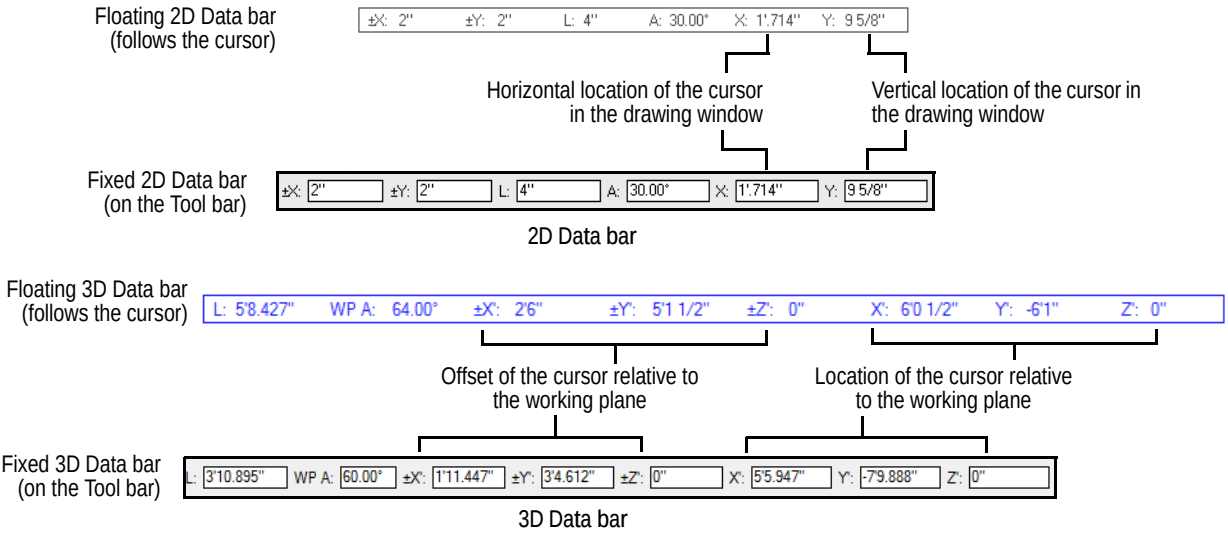
Source Object or Attribute	Result
Object, such as a rectangle, with a rotated creation mode	Matches rotated/non-rotated creation mode of source object
Symbols	Makes the source symbol the active symbol, and activates the appropriate symbol insertion tool
Plug-in object	Sets the tool parameters to those of the source plug-in object
Wall	Matches the wall style of the source wall

3. Create the similar object in the same way the source object was created.

   Because the Attributes palette may have changed to match the source object, the Attributes palette utility menu can quickly set or change attributes globally if needed (see “Setting Global and Default Attributes” on page 1094).

Using the Data Bar

Use the Data bar when you draw to lock certain values for an object, such as the length of a circle radius, the angle of a wall, the length of a hemisphere radius, or the height of an extruded rectangle. The fields that are available on the Data bar depend on the active tool and on the action being performed.



The data displayed in the bar is gathered from the feedback segment of the object being created. The coordinates are relative to a floating datum, if one exists; otherwise the coordinates are absolute. The more common Data bar fields are described in the following table; other fields that can display are described where their use is relevant.

Click to show/hide the parameters.

Parameter	Description
2D Parameters	
Delta X	The offset distance from the previous click or position along the X axis
Delta Y	The offset distance from the previous click or position along the Y axis
L	The length or distance from 0,0, or the radius of the object that is being drawn
A	The angle or rotation of the object that is being drawn

Parameter	Description
X	The X (horizontal) location of the cursor in the drawing window
Y	The Y (vertical) location of the cursor in the drawing window
3D Parameters	
WP A	The angle or rotation of the object that is being drawn relative to the working plane X axis
X’	The absolute location of X’ in working plane coordinates
Y’	The absolute location of Y’ in working plane coordinates
Z’	The absolute location of Z’ in working plane coordinates (applies to 3D objects only)
L	The length or distance of the object that is being drawn
Z	The Z (depth) of the object being drawn
Ctr X	The center of the object along the X axis
Ctr Y	The center of the object along the Y axis
Ctr Z	The center of the object along the Z axis
Delta X’	The offset distance from the previous click or position of X’ in working plane space
Delta Y’	The offset distance from the previous click or position of Y’ in working plane space
Delta Z’	The offset distance from the previous click or position of Z’ in working plane space
Delta X	The offset distance from the previous click or position of X
Delta Y	The offset distance from the previous click or position of Y
Delta Z	The offset distance from the previous click or position of Z
Radius	The radius of the object being drawn
Height	The height of the object being drawn

The location and behavior of the Data bar fields can be adjusted by options listed under **Windows > Data Bar Options** or from the Data Bar and Edit Group Options list (see “Data Bar and Edit Group Options” on page 127 for details).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Drawing with the Data Bar
Data Bar and Edit Group Options

Drawing with the Data Bar

To draw an object with the Data bar:

1. There are two different ways to activate the Data bar for drawing.
   - Select a drawing tool and click once to begin to draw the object. Press the Tab key to activate the first field in the Data bar, or press Shift+Tab to activate the last field. Alternatively, if the Data bar options are set so that instant activation is enabled, type the value for the first field in the Data bar to activate it.
   - Select a drawing tool and press Tab to activate the first field in the Data bar. After entering the data, click to begin drawing.

   In click-drag drawing or moving operations, after you press Tab or Shift+Tab to activate the Data bar field, you can release the mouse button for ease of data entry. Complete the operation with a mouse click or press the Enter key twice.
2. Enter values in the appropriate field(s), using the keys as follows.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

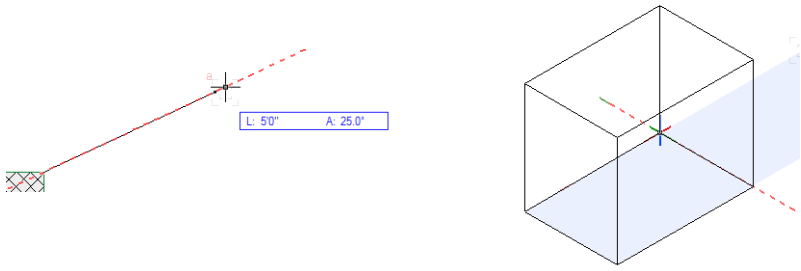
Key	Action
Enter	• When the focus is in a Data bar field, sets the value that is currently displayed, and moves the focus to the drawing area • When the focus is in the drawing area, completes the object (or completes the current segment of the object, for path objects such as polygons, walls, and dimensions)
Tab	• When the focus is in a Data bar field, sets the entered value and moves to the next field (if no value was entered, the field is not set) • When the focus is in the drawing area, moves the focus to the first field in the Data bar
Shift+Tab	• When the focus is in a Data bar field, sets the entered value and moves to the previous field (if no value was entered, the field is not set) • When the focus is in the drawing area, moves the focus to the last field in the Data bar

To clear an entry before it is set, press the Backspace key. The previous value in that field redisplays.

If the SmartCursor option is enabled in Vectorworks preferences, a dotted line displays to represent the location of the values entered for the X, Y, and Z axes.

3. A vector lock (red dashed line) displays to represent the location of the values entered in the Data bar.

   See “Creating Vector Locks” on page 147 for more information.

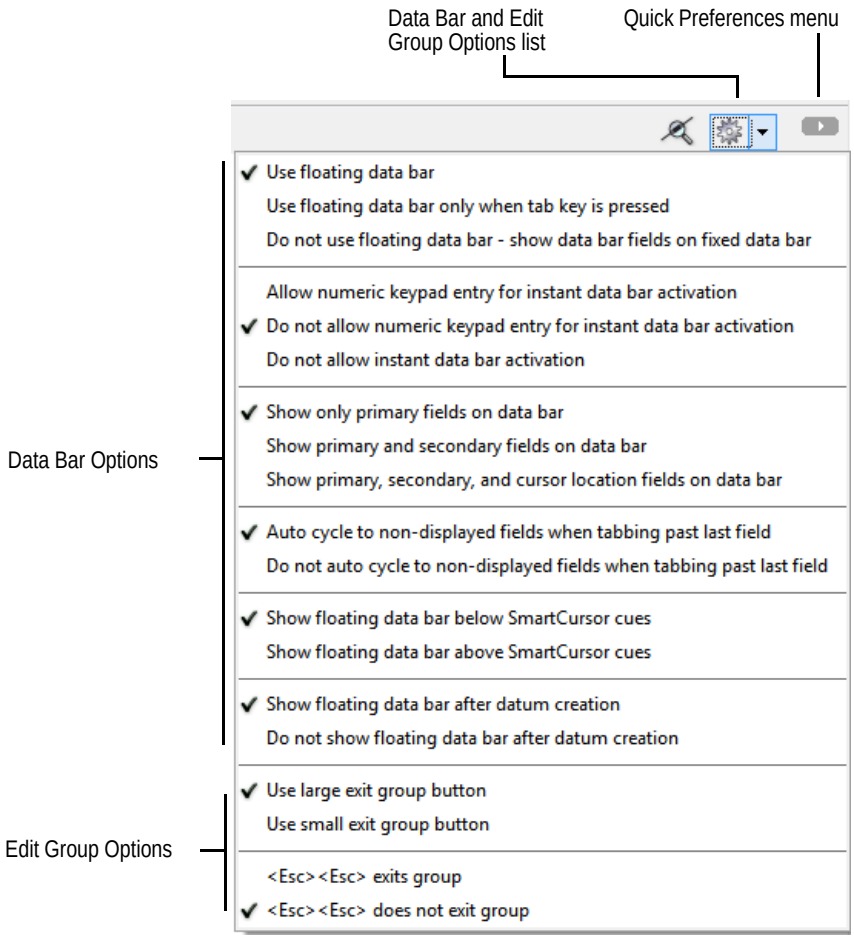


4. To complete the object according to the values you entered, click the mouse button. For non-path objects, you can also press Enter to complete the object if the focus is in the drawing area, or press Enter twice to complete the object if the focus is in the Data bar. For path objects, such as polygons, walls, and dimensions, you may need to click to complete the object.

Using the Data Bar
Data Bar and Edit Group Options

Data Bar and Edit Group Options

The Data bar is controlled by options on a drop-down list on the right side of the Tool bar. Options for editing groups are also provided in this list. The Data Bar and Edit Group Options list is enabled by default on the **Quick Preferences** menu on the Tool bar; therefore, it displays as a button to the left of the **Quick Preferences** menu. The **Data Bar Options** and **Edit Group Options** can also be selected from the **Window** menu.



Menu Command	Description
Location of the Data bar fields	
Use floating Data bar	The Data bar floats with the cursor in the drawing area
Use floating Data bar only when tab key is pressed	The Data bar floats with the cursor only when the Tab key is pressed; otherwise the bar does not display
Do not use floating data bar - show data bar fields on fixed data bar	The Data bar displays on the top left side of the window, at the top of the Tool bar
Activation of the Data bar	
Allow numeric keypad entry for instant Data bar activation	When the floating Data bar is displayed, enter numbers on the keyboard or numeric keypad to activate the first field
Do not allow numeric keypad entry for instant Data bar activation	When the floating Data bar is displayed, enter numbers on the keyboard only to activate the first field
Do not allow instant Data bar activation	When the floating Data bar is displayed, press the Tab key to activate the first field
Field display on the Data bar	
Show only primary fields on Data bar	Show only the primary data fields (for example, L and A for circles)
Show primary and secondary fields on data bar	Show all fields except the cursor location fields (X and Y)

Menu Command	Description
Show primary secondary, and cursor location fields on data bar	Show all fields
Field cycling options	
Auto cycle to non-displayed fields when tabbing past last field	When some of the data fields are hidden, press the Tab key in the last visible field to make the non-displayed fields become visible
Do not auto cycle to non-displayed fields when tabbing past last field	When some of the data fields are hidden, press the Tab key in the last visible field to return to the first field
Location of the floating Data bar	
Show floating Data bar below SmartCursor cues	When the floating Data bar is enabled, displays the Data bar below the SmartCursor cues (below the cursor)
Show floating Data bar above SmartCursor cues	When the floating Data bar is enabled, displays the Data bar above the SmartCursor cues (above the cursor)
Data bar activation on datum creation	
Show floating Data bar after datum creation	When the floating Data bar is enabled, displays the Data bar automatically after a datum is set
Do not show floating Data bar after datum creation	The floating Data bar does not automatically display after a datum is set. Press the Tab key to display the floating Data bar.
Display of the Exit Group button	
Use large exit group button	When a group is being edited, show a large button with the label **Exit Group** in the upper right corner of the drawing area
Use small exit group button	When a group is being edited, show a small button with an arrow icon in the upper right corner of the drawing area
Exit group options	
<Esc><Esc> exits group	When a group is being edited, pressing the Escape key twice exits the editing window
<Esc><Esc> does not exit group	When a group is being edited, pressing the Escape key twice does not exit the editing window

Using the Data Bar
Drawing with the Data Bar
Setting Quick Preferences

Drawing with Snapping

SmartCursor snapping allows you to draw with precision in both 2D and 3D. Used in conjunction with drawing tools, the SmartCursor uses snap indicators, hints, extension lines, text (SmartCursor cues), and sounds, and then precisely snaps the cursor based on the snapping categories enabled in the Snapping palette.

Setting Snapping Parameters

The Snapping palette manages SmartCursor snapping; the on-screen feedback which displays while drawing is based on the snapping categories enabled in the Snapping palette. Different drawing tasks may require a variety of snapping combinations. A snapping category is enabled by clicking the desired tool from the Snapping palette. Each tool toggles to turn the snapping category on or off.

Snapping categories can be used individually or combined for precise drawing and snapping. For example, snap to object points and snap to the grid to find points which are both on an object and on a grid line. Snapping can also be combined with Data bar input to find a specific snap point along a defined location in the drawing area.

Snapping is available for both 2D and 3D tools; however, Snap to Tangent only applies to 2D tools, and Snap to Working Plane only applies to 3D tools.

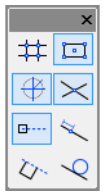
The snapping parameters of most of the snapping tools are set in the SmartCursor Settings dialog box. Most of these settings are considered application settings, and apply to any file opened in the Vectorworks program. Certain settings are considered document settings and are saved with the file; these include grid dimensions, alternate angle extension lines, datum and smart edge offset, and distance snap settings. Some of the snapping tools do not have additional parameters, and are simply toggled on or off.

When drawing, it may become necessary to temporarily suspend SmartCursor snapping. Press and hold the ‘ key (back quote key) to suspend all snapping. The snapping settings are retained, and snapping resumes when the ‘ key is no longer pressed. The shortcut key for suspending snapping can be customized; see “Modifying Snapping and Mode Shortcuts” on page 1842.

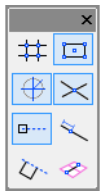
To set up and activate snapping categories:

1. If the Snapping palette is not open, select **Window > Palettes > Snapping**.

   The Snapping palette opens. The last tool on the palette is different depending on whether a 2D or 3D tool is currently active.



2D Snapping palette



3D Snapping palette

Snapping suspended with back quote key

2. Double-click a snapping tool to open the SmartCursor Settings dialog box with specific snapping parameters for that tool. Some of the tools do not have any additional parameters.

   The SmartCursor Settings dialog box can also be opened by selecting **Tools > SmartCursor Settings**. Each tab of the dialog box is described in the following sections.

3. Enter the desired criteria, and then click **OK** to set.

   The criteria are described in the following sections.

4. Click a snapping tool to activate it.

   Snapping palette shortcut keys can be set in the Workspace Editor; see “Modifying Snapping and Mode Shortcuts” on page 1842.

   Different combinations of constraints and snapping parameters are required for different drawing tasks. Select the **Tools > Custom Tool/Attribute** command to easily create a script with the current SmartCursor settings and active tools on the Snapping palette. Execute the script by double-clicking on it from the Resource Browser to quickly change snapping parameters and settings. See “Creating Custom Tool/Attribute Scripts” on page 1773 for more information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

General Snapping

The parameters on the General tab specify global SmartCursor functionality.

To set general snapping parameters:

1. Select **Tools > SmartCursor Settings**, or double-click a Snapping palette tool that has additional parameters.

   The SmartCursor Settings dialog box opens. Click the General tab.

Click to show/hide the parameters.

Parameter	Description
Show SmartCursor Cues	Displays SmartCursor cues while drawing. The cues displayed depend on the combination enabled in the Snapping palette, as well as the current cursor position.
Show Snap Points	Displays temporary graphical hints to indicate that a snap point is available; see “Snapping Indicators” on page 144 for more information. Deselect to hide snap points.
Zoom Line Thicknesses in Snap Loupe	When using the snap loupe (see “Using the Snap Loupe” on page 148), sets the line thicknesses relative to the zoom level so that objects display normally in the loupe view
Snap to Combined Page Area	Snaps to the edges and center of the combined printable area of design layer and sheet layer pages
Snap to Individual Pages	Snaps to the edges and center of the individual pages within design layer and sheet layer pages
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Ignore Non-planar Snaps in Planar Contexts	Snaps only to snap points that are planar within the current plane's context, ignoring snap points that are not co-planar. When using the automatic working plane with a tool that operates on a plane, the current planar context is the plane that is acquired by the automatic working plane.When not using the automatic working plane, and you are creating a 2D object, the current planar context is the current working plane. This is useful in Wireframe mode when the automatic working plane is on, and the cursor is over a planar face (not a curved face); and when the automatic working plane is off, and you are drawing planar objects. It is not needed in rendered views because snapping does not occur for geometry hidden behind solid objects.
Interactive Appearance Settings	Opens a dialog box to change the appearance of the interactive elements in the Vectorworks program, including general elements, object highlighting, SmartCursor elements, and snap points; see "Configuring Interactive Display" on page 116
Note	Reminds you that snapping can be suspended temporarily by pressing the ‘ key (back quote key)
Reset	Restores default parameter settings for all tabs

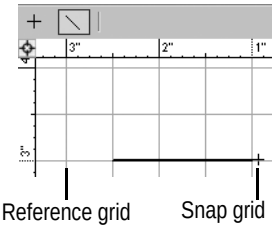
2. Click **OK**.

SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping
The Automatic Working Plane

Grid Snapping

The snap and reference grids assist with precise drawing on both design and sheet layers, as described in "Snap and Reference Grids" on page 71.

When snap to grid is on, the SmartCursor snaps to set points on the snap grid. For example, if the grid is set to 1", as the mouse moves over the grid, it automatically "catches" every inch. When you create a line, the line's first and last point will (if no other snapping is active) lie on the snap grid. Snap to grid is the only type of snapping that does not provide any sort of visual cues. If snap to grid is on, the mouse is always on the grid, unless other snaps are also selected which override snap to grid.



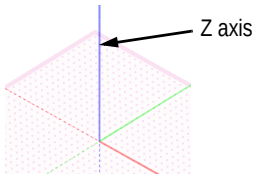
When snap to grid is on, the working plane displays with "dots." The dots do not display for working planes when snap to grid is off. See "Appearance of the Planes" on page 1169.

To set grid snapping:

1. Select **Tools > SmartCursor Settings**, or double-click the **Snap to Grid** tool in the Snapping palette.

   The SmartCursor Settings dialog box opens. From the Grid tab, specify the snap and reference grid settings.

Click to show/hide the parameters.

Parameter	Description
Snap Grid	The grid that aligns (snaps) objects during drawing and editing, when Snap to Grid is active
Symmetrical	Select to constrain the Y setting to be the same as the X setting, resulting in a symmetrical grid (this setting is selected by default)
X and Y	Sets the grid dimensions in the X and Y directions; the Y setting appears dimmed if **Symmetrical** is selected
Reference Grid	The visible grid used for visually aligning objects during drawing and editing
Symmetrical	Select to constrain the Y setting to be the same as the X setting, resulting in a symmetrical grid (this setting is selected by default)
X and Y	Sets the grid dimensions in the X and Y direction; the Y setting appears dimmed if **Symmetrical** is selected
Grid Options	Sets how the reference grid behaves on screen and when printing
Show Grids	Displays the reference grid when possible (even with this option selected, the reference grid may not display depending on the zoom factor); also displays the active layer plane grid in 3D views. This setting is selected by default.
Print Grid	Prints the reference grid on both design layers and sheet layers
3D Grid Axes	
Show 3D Z Axis	Shows the Z axis for working planes (this setting is selected by default) 
Show 3D Axis Labels	Shows labels for each axis in the active layer plane and working plane

2. Click **OK**.
3. To activate snapping to the snap grid, click the **Snap to Grid** tool from the Snapping palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Snap and Reference Grids
SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

Object Snapping

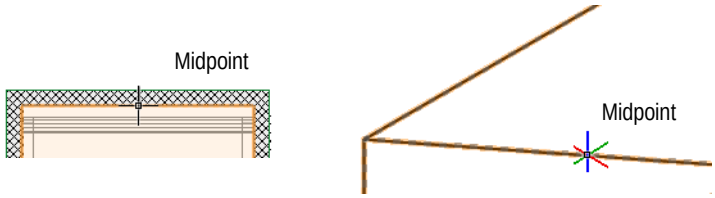
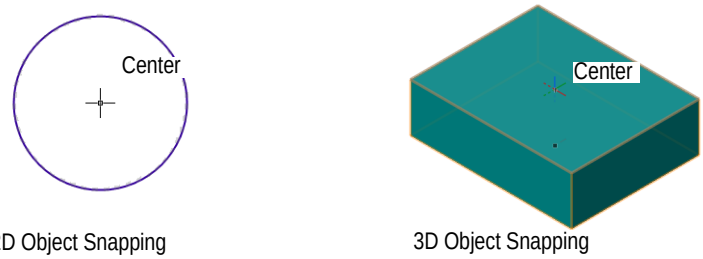
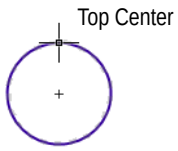
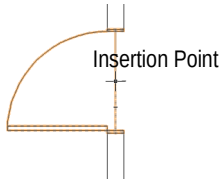
When snap to object is on, the SmartCursor finds specific parts of an object, such as corners, endpoints, midpoints, or centers of 2D objects and arc edges, as well as meshes, extrudes, sweeps, spheres, 3D polygons, 3D planar faces, and walls, floors, roofs, roof faces, loci, and columns. Cues display near the cursor to identify the location.

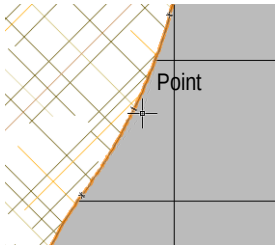
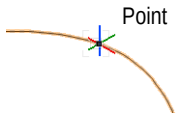
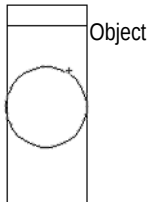
 To set object snapping:

1. Select **Tools > SmartCursor Settings**, or double-click the **Snap to Object** tool in the Snapping palette.

   The SmartCursor Settings dialog box opens. From the Object tab, specify the object snap settings.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Click to show/hide the parameters.

Parameter	Description
End Point	Finds the endpoint of arcs, and the end of an object's edge 
Mid Point of Edge	Finds the middle of the edge of an object 
Center Point	Locates the center of 2D objects and arc edges, as well as meshes, extrudes, sweeps, spheres, 3D polygons, 3D planar faces, and walls, floors, roofs, roof faces, and columns 
Quadrant Points	Finds the top, left, right, and bottom of circles, ovals, rectangles, and rounded rectangles 
Insertion Point	Locates the object origin of loci, lights, objects, symbols, text, and for dimensions, one of the referenced points 

Parameter	Description
Vertex	Locates the corner point on polyline, polyline Bézier, and polyline cubic segments; for 3D, finds the vertex of a solid or the control point of a NURBS curve  
Nearest Point on Edge	For 2D, finds the closest point on an object's edge, and for 3D, finds any non-specific point along a solid edge or NURBS curve. It may be more convenient to use smart edge snapping instead; see "Smart Edge Snapping" on page 140. 

2. Click **OK** to set the object snapping options.
3. To activate object snapping, click the **Snap to Object** tool from the Snapping palette.

SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

Angle Snapping

Two points define an angle; Snap to Angle only applies to the second point of a two-point feedback segment, such as when drawing a line or polygon. When snapping to angles, the SmartCursor finds the angles specified, and by default, the SmartCursor also finds angles relative to the horizontal and vertical axes. Snap to Angle can also detect a plan rotation angle (Vectorworks Design Series required); items created along that angle when in a rotated plan view, appear horizontal when in a non-rotated, world coordinate view.

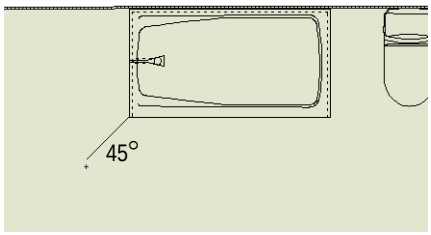
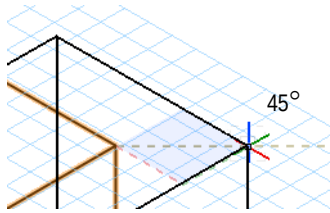
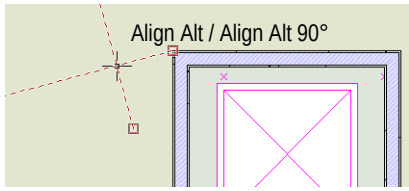
 To set angle snapping:

1. Select **Tools > SmartCursor Settings**, or double-click the **Snap to Angle** tool in the Snapping palette.

   The SmartCursor Settings dialog box opens. From the Angle tab, specify the angle snap settings.

Click to show/hide the parameters.

Parameter	Description
Angles	

Parameter	Description
Angles from Axes	Snaps to the specified angles with respect to the horizontal or vertical axes. For example, if 45° is specified, the angles found include 45°, 135° (90+45), 225°(180+45), and so on. Specify the angles to detect, separated by a semi-colon; 45° and 30° are suggested by default. Up to eight angles can be set.  
Angles relative to prior segment	Snaps to the specified angles relative to the previously drawn segment, instead of with respect to the axes. This is convenient when drawing multi-segment shapes, since the angle snap is based on the prior segment while drawing, and no mental arithmetic is required. The / key toggles this option on and off. When off, angles snap relative to the axes as in the **Angles from Axes** setting. The shortcut key can be set in the workspace editor; see “Modifying Snapping and Mode Shortcuts” on page 1842
Plan Rotation (Vectorworks Design Series required)	Finds the angle of plan rotation; this snap is useful when drawing in a rotated top/plan view, and requiring objects to be horizontal when the plan is no longer rotated
Alternate Coordinate System	Specifies an additional rotated coordinate system that provides angle snaps and alignments along its axes
Enter Angle	Sets an angle other than the angles entered in **Angles from Axes**. Enter an alternate angle.
Extension Lines	Draws an extension line from a smart point to the alternate angle and 90° to the alternate angle 

Holding the Shift key while drawing forces the SmartCursor to select the nearest snap angle.

2. Click **OK** to set the angle snapping options.
3. To activate angle snapping, click the **Snap to Angle** tool from the Snapping palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

Smart Points Snapping

When smart points snapping is on, you can temporarily remember a point where the cursor paused for a set length of time, or where a special shortcut key was pressed. Once a smart point has been defined, you can align to it horizontally, vertically, or at a specific angle using extension lines and SmartCursor cues.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To set a smart point, pause the cursor over an object point for the set number of seconds, or press the T key. A small box displays when the smart point has been set.

A special smart point, called a datum, can be set to temporarily create a new origin for snapping and measuring purposes. The datum does not need to be set over an object, but can be placed anywhere. When a datum is set, all measurements along the X and Y axis are taken from that point, rather than the drawing's origin, until the datum is moved or deactivated. When a datum is created, the Data bar displays coordinates relative to the datum, using the datum point as the origin. If a datum is set before moving an object with the **Selection** tool, the object's location and angle are measured relative to the datum.

To set a datum point, pause the cursor for the set number of seconds, or press the G key. The datum displays as a small circle around the chosen point. See "Data Bar and Edit Group Options" on page 127 to activate the floating Data bar automatically after a datum is set.

The SmartCursor remembers up to three smart points, and then the oldest points are replaced. A smart point or datum can be released by pausing the cursor or pressing the shortcut key over the point again.

Press the Esc key to clear all smart points.

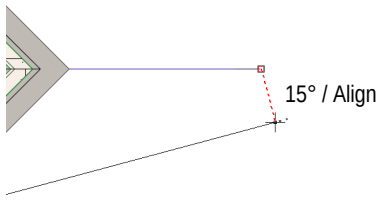
A current snap is needed to acquire smart points, so enabling Snap to Object, at a minimum, is recommended.

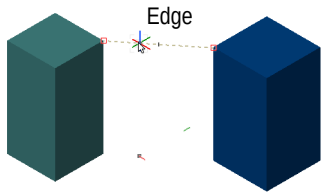
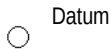
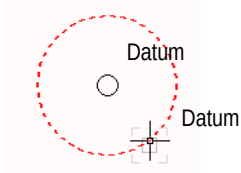
To set smart point snapping:

1. Select **Tools > SmartCursor Settings**, or double-click the **Smart Points** tool in the Snapping palette.

   The SmartCursor Settings dialog box opens. From the Smart Point tab, specify the smart point snap settings.

Click to show/hide the parameters.

Parameter	Description
Horizontal/Vertical Extensions	Displays an extension line when the cursor aligns horizontally, vertically, or in the X, Y, or Z direction with a smart point or datum 
Extension Lines	Creates an extension line from a smart point when drawing at a snap angle and the SmartCursor is aligned perpendicular to the smart point 

Parameter	Description
Snap to line between Smart Points	Snaps to the extension line between smart points 
Allow extensions from nearby Smart Points	Creates extension lines from smart points located near the cursor. When deselected, prevents extension lines generated by smart points located close to the cursor, so that alignment does not occur to unintentionally acquired smart points in cluttered drawings.
Acquire Smart Point	Sets a smart point at the cursor location when the cursor pauses for the indicated number of seconds. (Experienced users may wish to reduce the pause time to 0.1 or 0.2 seconds.) Alternatively, press the T key to set or release a smart point at the cursor location, even if the **Acquire Smart Point** option is disabled. 
Set Datum	Sets a floating datum to define a temporary data origin by pausing the cursor for the indicated number of seconds (normally, set this time interval to be longer than the smart point acquisition time). Alternatively, press the G key to set or release a datum at the cursor location, even if the **Set Datum** option is disabled. 
Datum Offset	Sets an offset from the temporary datum origin that is indicated by an additional marker along the extension line; enter the offset value 

2. Click **OK** to set the smart point options.
3. To activate smart point snapping, click the **Smart Points** tool from the Snapping palette.

   The appearance of the smart points can be changed; see “Configuring Interactive Display” on page 116. The shortcut key assigned to smart points and the datum can be customized; see “Modifying Snapping and Mode Shortcuts” on page 1842.

SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

Smart Edge Snapping

When smart edge snapping is on, the SmartCursor finds points on, or at a certain distance away from, a specified edge. The edges of linear 2D geometry, arc and polyline curve segments, and 3D segments and curves, can be acquired as smart edges. Once a smart edge has been defined, you can align to it horizontally, vertically, or at a specific angle using extension lines and SmartCursor cues. A smart edge also provides snapping to a point on an edge without having to activate **Nearest Point on Edge** object snapping.

To set a smart edge, slowly move the cursor over a 2D object edge for the set number of seconds, or press the T key while over an edge. Dotted extension lines indicate that the smart edge has been set.

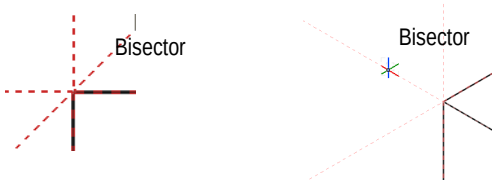
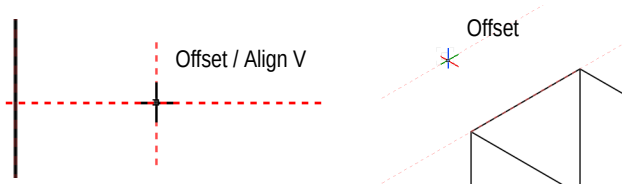
Up to two smart edges can be set, and then the oldest edges are replaced. A smart edge can be released by moving the cursor slowly over the edge or pressing the T key over the edge again.

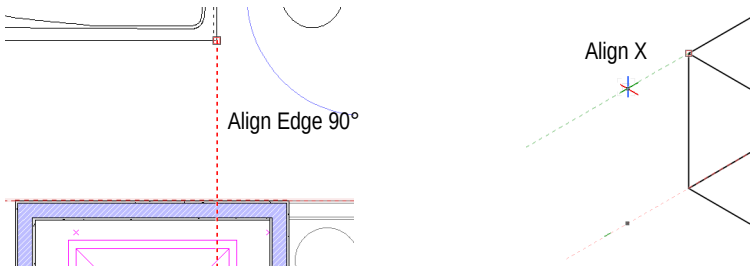
 To set smart edge snapping:

1. Select **Tools > SmartCursor Settings**, or double-click the **Smart Edge** tool in the Snapping palette.

   The SmartCursor Settings dialog box opens. From the Smart Edge tab, specify the smart edge snap settings.

Click to show/hide the parameters.

Parameter	Description
Acquire Edge	Sets a smart edge at the cursor location when the cursor moves over an object edge for the indicated number of seconds. Alternatively, press the T key to set or release a smart edge at the cursor location, even if the **Acquire Edge** option is disabled. 
Snap to Bisector	Finds points along the bisector that intersects two smart edges 
Snap to Offset	Sets an offset from either end of a smart edge extension line that is indicated by additional markers along the extension line; enter the offset value 

Parameter	Description
Snap to Extension Lines	Generates extension lines from smart points perpendicular and parallel to the smart edge 

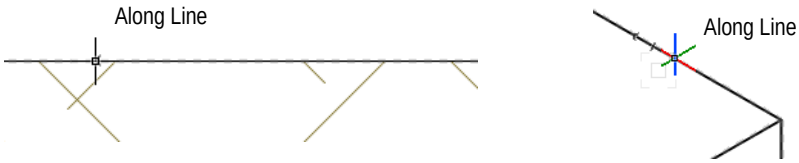
2. Click **OK** to set the smart edge options.
3. To activate smart edge snapping, click the **Smart Edge** tool from the Snapping palette.

   The appearance of the smart edge can be changed; see “Configuring Interactive Display” on page 116. The shortcut key assigned to the smart edge can be customized; see “Modifying Snapping and Mode Shortcuts” on page 1842.

SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

Distance Snapping

When snap to distance snapping is on, the SmartCursor finds points at a selected distance along a straight or curved line, polygon edges, wall edges, and other linear objects.



 To set the snap distance:

1. Select **Tools > SmartCursor Settings**, or double-click the **Snap to Distance** tool in the Snapping palette.

   The SmartCursor Settings dialog box opens. From the Distance tab, specify the distance snap settings.
2. Set the distance using a fraction, percentage, or a dimensional distance. Select **Multiple Divisions** to repeat snap points along a line. For example, the SmartCursor can snap every quarter inch, or every 1/8 of the length of the line.

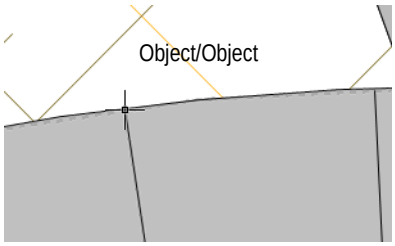
   The SmartCursor measures from each endpoint to the center of the line. If a line is 10 units long and the distance is set to 6 units, the distance will not be found at all, since it is longer than half a line. Similarly, any fraction greater than 1/2 the length of the object or less than 0 cannot be used.
3. Click **OK** to set the Snap to Distance options.
4. To activate Distance snapping, click the **Snap to Distance** tool from the Snapping palette.

SmartCursor Cues
Setting Snapping Parameters

Drawing with Snapping

Snap to Intersection

When snap to intersection snapping is on, the SmartCursor finds the intersection between two objects or between the parts of an object.



 To activate snapping to intersection:

Click the **Snap to Intersection** tool from the Snapping palette. No parameters are required.

SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

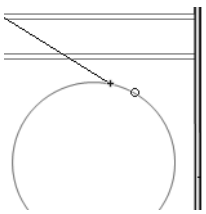
Snap to Tangent (2D Only)

The Snap to Tangent option uses the SmartCursor to locate tangents on circular arc geometry when drawing.

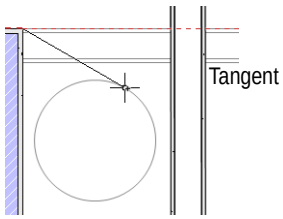
Tangents cannot be found on quarter arcs.

 To activate tangent snapping:

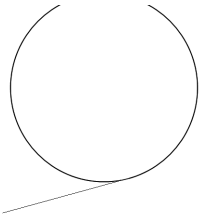
Click the **Snap to Tangent** tool from the Snapping palette. No parameters are required.



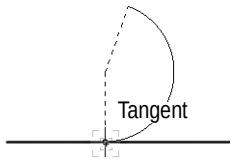
While drawing, as the feedback segment approaches a tangent, the special snap points indicator shows that a tangent snap is available



Move the cursor to the tangent snap, and click to create a line that is tangent to the circle



Alternatively, begin drawing a line from the arc geometry, and the feedback segment remains tangent to the circle. Click to create a tangent line.



Use tangent snapping to find the tangent when drawing arcs or polylines in Tangent Arc mode

Hold down the Option (Mac) or Alt (Windows) key to switch the tangent to the opposite side of the object.

Click here for a video tip on this topic (Internet connection required).

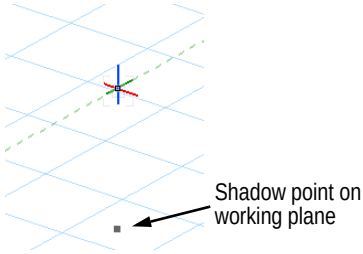
SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

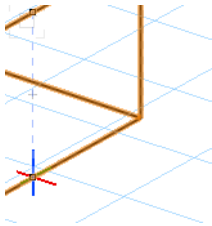
Snap to Working Plane (3D Only)

When Snap to Working Plane snapping is on, the SmartCursor snaps/projects any point not on the working plane down to its shadow point on the working plane.

To activate snapping to the working plane:

Click the **Snap to Working Plane** tool from the Snapping palette. No parameters are required.





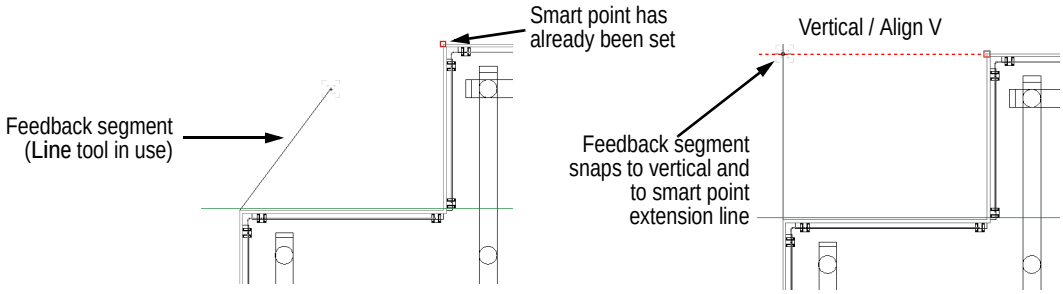
While moving along the top edge of the extrude with working plane snapping on, the cursor snaps to the working plane

SmartCursor Cues
Setting Snapping Parameters
Drawing with Snapping

Snapping Techniques

A variety of settings and options combine to facilitate drawing with snapping. Several types of snap types and snapping indicators assist with snapping. The snap loupe temporarily zooms into the drawing view near the cursor to obtain a snap in a complex drawing.

Snapping is used with the feedback segment preview line that displays while drawing with most tools. For example, when drawing a line, you may want it to be vertical, and also aligned with a point on your drawing. To do this, set a smart point (with extension lines) on the point to align to. Angle snapping should be enabled (because it snaps to the vertical) along with smart point snapping. The feedback segment of the line snaps to the vertical and to the smart point, and a second click finishes drawing the line as desired.



Several areas in the program control the various aspects of snapping.

Functionality	Location/Description
Enable/disable snapping tools	Snapping palette
Enable/disable individual snapping parameters	SmartCursor Settings dialog box
Change the appearance of snap points and indicators	Interactive Appearance Settings dialog box; “Configuring Interactive Display” on page 116

Functionality	Location/Description
Enable/disable snap box, selection box, and acquisition hints	Interactive tab of Vectorworks preferences
Change the appearance of the snap box and selection box	Interactive tab of Vectorworks preferences
Save snap settings as a script	**Custom Tool/Attribute** command; “Creating Custom Tool/Attribute Scripts” on page 1773
Change snapping shortcut keys	Keys tab of the Workspace Editor; “Modifying Snapping and Mode Shortcuts” on page 1842
Snapping to objects in other layers or in classes or layers set to gray	Class and layer option commands; “Setting Class and Design Layer Options” on page 193

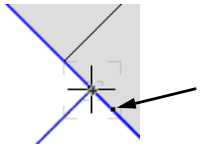
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Snapping Indicators

There are several types of snap points. Use these in combination with the settings in the Snapping palette, SmartCursor Settings dialog box, the Interactive Appearance Settings dialog box, and Interactive tab in Vectorworks preferences to tailor snapping and snapping appearance to your drawing task. In addition to the snap points, other interactive elements assist with snapping.

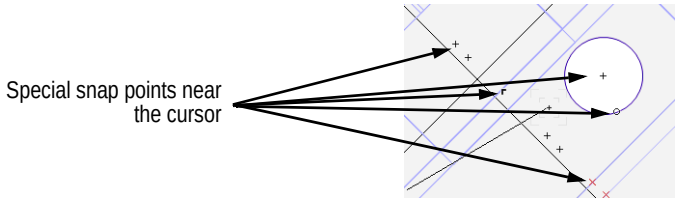
- Available snap points

  Available snap points display near the cursor, within the snap box area, to indicate that a snap point is near the cursor.



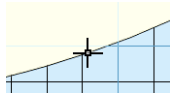
- Special snap points

  Special snap points display in the general cursor area, to inform you that midpoint, center, along line, and/or tangent snap points are nearby. The visible snap points depend on the snapping enabled in both the SmartCursor Settings dialog box and the Snapping palette.



- Current snap point

  The current snap point shows that the SmartCursor is snapped.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



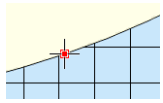
The current snap point indicator is black in 2D. In 3D, the indicator displays with red, green, and blue colors, which correspond to the red (X), green (Y), and blue (Z) axes. The orientation of the 3D indicator is relative to the current working plane; dashed lines indicate the negative axes.

2D current point

3D current point

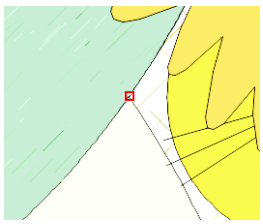
- Obtained snap point

  When drawing with a tool, such as the **Line** tool, the first click of the tool on a current snap becomes the obtained snap point. The obtained snap point only displays for a brief moment, letting you know that the click was snapped.

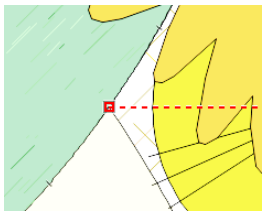


- Smart point

  When the Smart Point snapping is enabled, up to three smart points can be defined at one time; see “Smart Points Snapping” on page 137. A smart point displays as a red rectangle by default. If extension lines are enabled, the extension line for the smart point appears as a dashed line. A datum is a special type of smart point and counts as one of the three smart points.

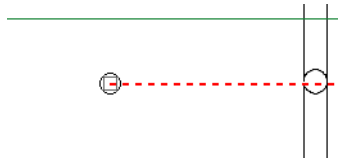


Smart point



Smart point with extension line

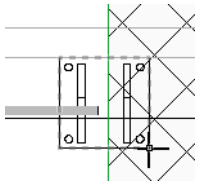
Datum



Datum with extension line

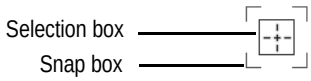
- Snap object highlighting

  The object from which the snap points are generated is highlighted with a dashed line. This ensures that the correct snap points are being obtained within a congested drawing.



- SmartCursor selection box and snap box

  Two areas around the cursor assist with drawing: the selection box and snap box.

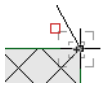
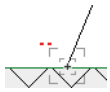


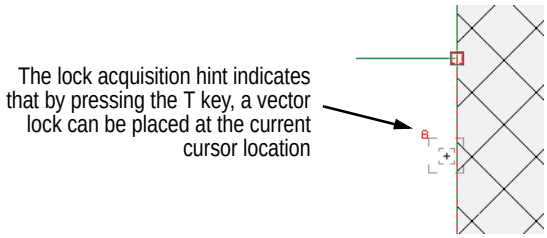
The selection box shows the area where object geometry can be selected or recognized by a tool that selects or clicks objects (such as the **Selection** tool or the **Eyedropper** tool). The snap box indicates the area from which the current snap will be obtained. The current snap is obtained from all the available snaps inside the snap box. The snap box allows a snap to be held while the selection box moves around to change the snapping selection.

The size and visibility of the selection box and snap box can be adjusted, and these indicators enabled, from the Interactive tab in Vectorworks preferences.

- Acquisition hints

  Acquisition hints display the potential snap points that can be acquired near the cursor. They indicate that a smart point, smart edge, or vector lock can be acquired.

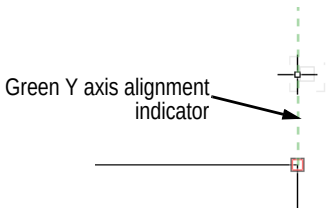
Acquisition Hint	Appearance
Smart point	
Smart edge	
Vector lock	

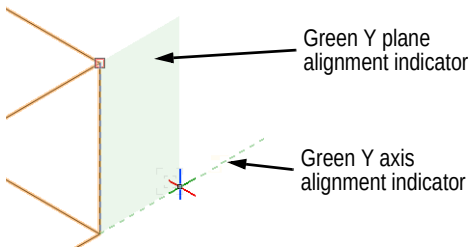


Enable the acquisition hints from the Interactive tab in Vectorworks preferences.

- Alignment Indicators

  Lightly colored axis and plane indicators provide subtle hints to help determine whether the cursor is aligned to an axis (2D) or an axis and/or plane (3D). The color of the indicator matches the axis color (red for X, green for Y, and blue for Z) to help you determine when the cursor is aligned to an axis or plane.





Setting Snapping Parameters
Drawing with Snapping
Interactive Preferences
Configuring Interactive Display

Creating Vector Locks

An extension line created from a smart point indicates a vector, or direction. This vector can be locked to facilitate drawing and moving objects, because the SmartCursor stays locked to that vector and serves as a guide for snapping along that vector.

To create a vector lock:

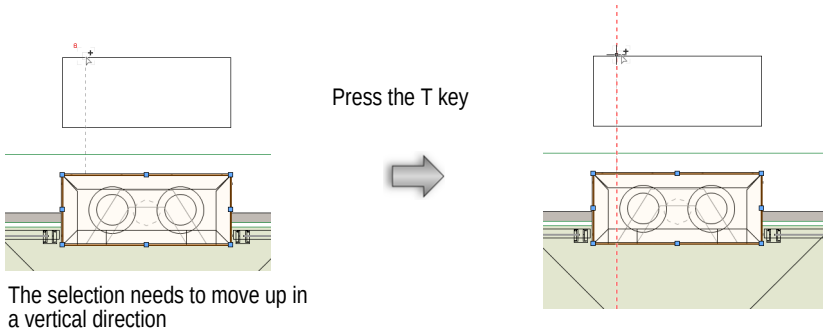
1. Select the tool for drawing, an object to move, or other operation that requires a vector lock.

   It is helpful to display acquisition hints, which show when a vector lock can be set. This is a Vectorworks interactive preference; see “Interactive Preferences” on page 56.

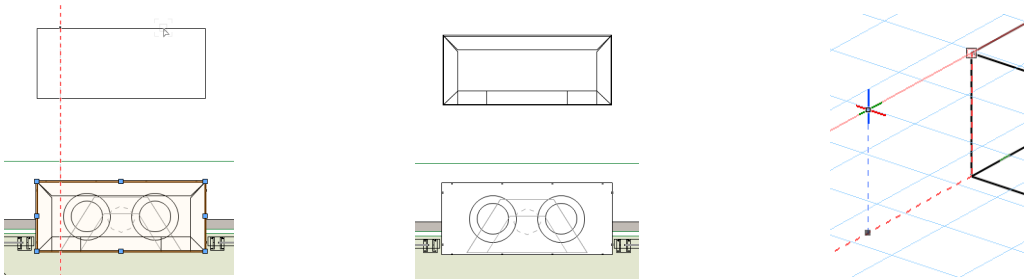
2. During the operation, the feedback segment indicates the vector direction and the acquisition hint at the cursor indicates that a vector lock can be set. A vector lock can be set along a smart point extension line. As the cursor moves along the extension line, the lock acquisition hint shows that a vector lock can be set.

3. Press the T key to create the vector lock in the direction of the extension line, or press the G key to set the vector lock relative to a datum, using the Data bar.

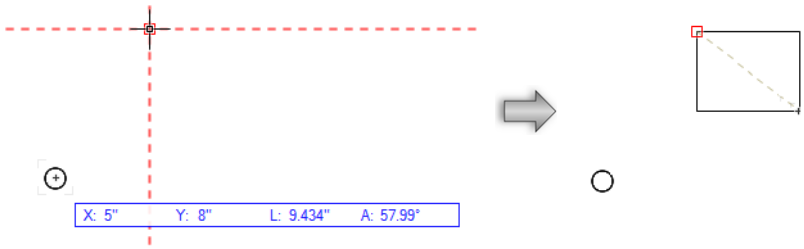
   The vector lock displays. The appearance of vector locks can be set as described in “Configuring Interactive Display” on page 116.



4. The SmartCursor locks to the vector and the operation is snapped along the desired direction.



The Data bar can create a vector lock along a specific direction. Enter values in the Data bar as described in “Using the Data Bar” on page 125. A vector lock displays, and the SmartCursor only snaps to points on the locked line.



Setting Snapping Parameters
Drawing with Snapping

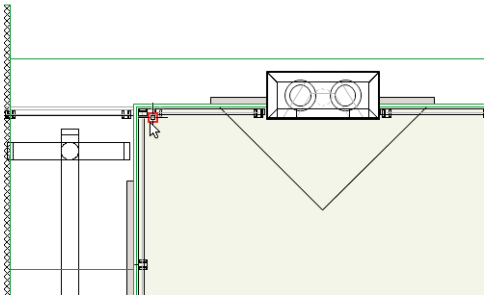
Using the Snap Loupe

In a congested drawing, it can be difficult to obtain the desired snap point. The snap loupe temporarily zooms into a drawing to obtain the snap. In this zoomed in view, the snap points near the mouse are easier to see.

The snap loupe is not intended to include navigation within the loupe window. It is a convenient way obtaining snap points and quickly returning to the drawing.

To use the snap loupe:

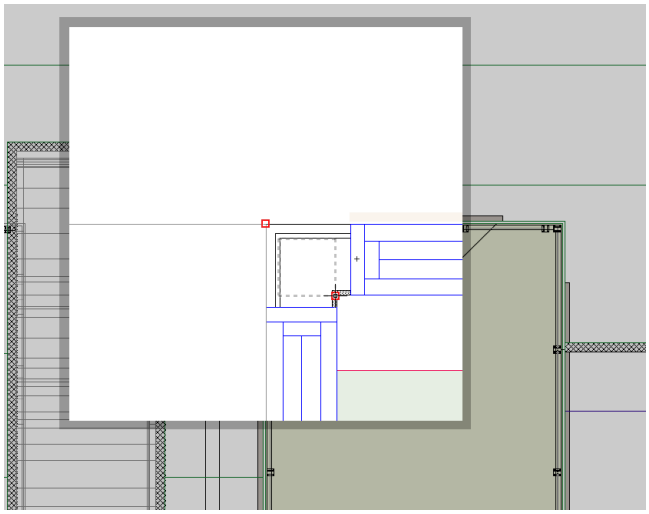
1. Move the mouse to the area where snapping is desired, but difficult.



2. Press the Z key.

   The snap loupe window opens, displaying the region near the cursor in a zoomed view. Within the loupe window, move the cursor to locate the desired point. If the snap loupe window is not sufficiently magnified to find a snap, press the Z key again.

   The **Zoom Line Thickness in Snap Loupe** preference determines how the drawing displays within the snap loupe. See “General Snapping” on page 132 to change this setting.



3. Click to perform the desired operation; the snap loupe window closes automatically, returning to the drawing.

   Alternatively, press the Esc key to exit the snap loupe without clicking, or click outside of the loupe to close it.

   Click here for a video tip about this topic (Internet access required).

Setting Snapping Parameters
Drawing with Snapping

SmartCursor Cues

The following table lists the individual cues that the SmartCursor uses along with a description of each. In many cases, two cues are used together to indicate that two snaps have been activated. For example, the cue ‘Align H/Angle’ means that the point located is both aligned horizontally to the indicated smart point and also snapping to an angle.

Some cues display when a first point has already been found, and a second point is being sought. These two points form a line called the feedback segment. This segment forms an angle and length which the SmartCursor uses for some of its snaps.

For a line, the feedback segment is the same as the line being drawn. However, for other objects, the interactive image does not lie on the feedback segment. The SmartCursor works on the feedback segment created from the first to the second points (for polygons, from the previous point to the current point).

Cue	Description
Snap to Angle	
Alt	Feedback segment is parallel to the alternative coordinate system angle
Alt 90°	Feedback segment is perpendicular to the alternative coordinate system angle
Delta angle	Feedback segment is at the specified snap angle from the rotated grid
Horizontal	Feedback segment is horizontal
Parallel	Feedback segment is parallel to an object
Perpendicular	Feedback segment is perpendicular to an object
Plan Rotation (Vectorworks Design Series required)	Feedback segment is aligned to the rotated plan angle
Symmetric	Feedback segment is sketching out a square or circle
Vertical	Feedback segment is vertical
X	Feedback segment is aligned with the X axis
Y	Feedback segment is aligned with the Y axis
Z	Feedback segment is aligned with the Z axis, or point is aligned to a smart point in the direction of the Z axis
X’	Feedback segment is aligned with the X axis of the rotated grid
Y’	Feedback segment is aligned with the Y axis of the rotated grid
Z’	Feedback segment is aligned with the Z axis of the rotated grid, or point is aligned to a smart point in the direction of the Z axis of the rotated grid
Smart Point	
----------	Extension line used for all smart point cues except Datum
Align	Feedback segment is perpendicular to a segment from the cursor to a smart point
Align H	Point is aligned horizontally with a smart point
Align V	Point is aligned vertically with a smart point

Cue	Description
Align X	Point is aligned with the X coordinate of a smart point
Align Y	Point is aligned with the Y coordinate of a smart point
Align Z	Point is aligned with the Z coordinate of a smart point
Align X’	Point is aligned with the X coordinate of a smart point, in the space of the rotated grid
Align Y’	Point is aligned with the Y coordinate of a smart point, in the space of the rotated grid
Align Z’	Point is aligned with the Z coordinate of a smart point, in the space of the rotated grid
Align Edge	Point is aligned to smart edge
Align Edge 90°	Point is aligned perpendicular to smart edge
Align Alt	Point is aligned to the user coordinate system
Align Alt 90°	Point is aligned perpendicular to the user coordinate system
Datum	Point is the Datum
O	Circle around Datum
Working Plane	Point is on the grid plane
Snap to Distance	
Along Line	2D—Point is at a specified distance along line 3D—Point is at a specified distance along line from the end point of a NURBS curve, 3D polygon, or edge of a solid object
Data bar	
----------	Extension line used to indicate X or Y value is locked
Angle	Feedback segment is constrained to an angle
Length	Feedback segment is of a fixed length set in Data bar
Snap to Object	
Arc	Cursor is over the corner point of a polyline arc segment
Arc Center	Cursor is over the center point of an arc
Arc End	Cursor is over the end of an arc segment
Bézier	Cursor is over the corner point of a polyline Bézier segment
Bottom Center	Cursor is over the bottom center of rectangle or group
Bottom Left	Cursor is over the bottom left of the object’s bounding box
Bottom Right	Cursor is over the bottom right of the object’s bounding box
Center	2D—Cursor is over the center of the object’s bounding box 3D—Cursor is over the geometric center of certain objects, NURBS curves, 3D polygons, or the center of a circular NURBS curve
Center Left	Cursor is over the center left of the object’s bounding box
Center Right	Cursor is over the center right of the object’s bounding box
Corner	Cursor is over the corner point of a polyline segment
Endpoint	Cursor is over the end of an object’s edge

Cue	Description
Fit	Cursor is over the corner point of a polyline cubic segment
Internal Origin	Cursor is over the file's internal origin marker
Insertion Point	Cursor is over the object origin for symbols, parametric objects, and text objects; for dimensions, the cursor is over one of the referenced points
Light	Cursor is on a light
Locus	Cursor is on a locus
3D Locus	Cursor is on a 3D locus
Midpoint	Cursor is over the middle of an object's edge
Object	2D—Cursor is on an object's edge 3D—Cursor is on any non-specific point along a solid edge or NURBS curve; displays at all corner points when the **Nearest Point on Edge** option is deselected for object snapping
Page	Cursor is over the page boundary
Page Bottom Center	Cursor is over the bottom center of the page boundary
Page Bottom Left	Cursor is over the bottom left of the page boundary
Page Bottom Right	Cursor is over the bottom right of the page boundary
Page Center	Cursor is over the center of the page
Page Center Left	Cursor is over the center left of the page boundary
Page Center Right	Cursor is over the center right of the page boundary
Page Top Center	Cursor is over the top center of the page boundary
Page Top Left	Cursor is over the top left of the page boundary
Page Top Right	Cursor is over the top right of the page boundary
Point	2D—Cursor is on an object point 3D—Cursor is on one of the vertices of a solid, the control point of a NURBS curve, the control point of a NURBS surface, or NURBS curve interpolation point
Top Center	Cursor is over the top center of the object's bounding box
Top Left	Cursor is over the top left of the object's bounding box
Top Right	Cursor is over the top right of the object's bounding box
Smart Edge	
Bisector	Point is on the bisector between two smart edges
Edge	Feedback segment is parallel to a smart edge, or cursor is on the edge between two smart points
Edge 90°	Feedback segment is perpendicular to a smart edge
Offset	Point is a preset distance from a smart edge
Smart Edge	Point is on a smart edge, but not on the object itself
Snap to Tangent	
Tangent	Feedback segment is tangent to an arc

Cue	Description
Tangent/Tangent	Feedback segment is tangent to two arcs

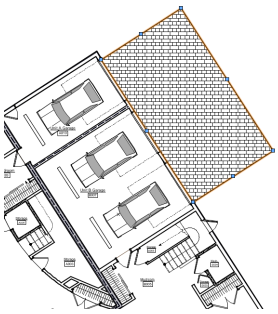
Setting Snapping Parameters
Drawing with Snapping

Vectorworks Modeling Environment

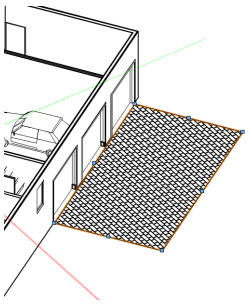
Vectorworks software provides robust 2D functionality with a variety of tools and commands for those who draft in Top/Plan. For those who design in 3D instead of, or in addition to, 2D, the Vectorworks program is a true modeling environment.

Planar Modes of 2D Objects: Screen Plane and Layer Plane

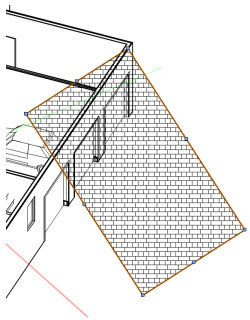
2D objects can be created as either screen plane objects or layer plane (planar) objects, depending on the selected planes mode. The screen plane is coincident to the computer screen, and perpendicular to the designer's view direction. 2D objects drawn in the screen plane "float" in 3D views. Planar objects, however, exist in 3D space; a planar object is drawn in a 3D plane—either the layer plane or a 3D working plane, maintaining its geometric relationship to the 3D model when in 3D views.



2D object in Top/Plan view



2D planar object in isometric view



2D screen plane object in isometric view

Objects are created as planar objects by default; they display in Wireframe mode with their graphic attributes on the layer plane in views other than Top/Plan, and become part of the 3D model. 2D objects can be created and edited on any plane and in any view; the editing occurs within the "planar context" of the object's plane. When performing operations on multiple selected planar objects in a 3D view, the objects must be co-planar.

Double-click on a planar object to set the working plane to the plane on which the object was created.

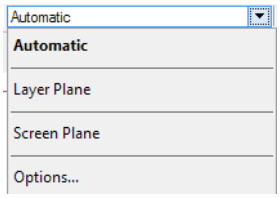
The **Split** tool, as well as the **Modify > Rotate > Rotate**, **Rotate Left,** and **Rotate Right**, and **Flip** commands, operate within the screen plane only.

While most 2D objects should normally be planar, some objects function best as screen plane objects. For example, all elements on sheet layers are, by definition, on the screen plane. Objects created as crop objects for viewports are automatically placed in the screen plane. In addition, when creating special hybrid symbols, the 2D component of the symbol should be created in the screen plane, so that the symbol has a representation when viewed in Top/Plan.

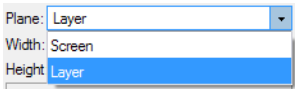
2D objects created in version 2010 and later retain their planar designation upon conversion. 2D objects created in versions of Vectorworks software prior to 2010 become screen plane objects upon conversion.

The Active Planes list located on the View bar sets the currently active plane, and determines whether 2D objects are created as planar or screen plane objects. The available planes depend on the drawing view, current tool, presence of named working planes, and plane mode options. To create planar objects, select **Layer Plane** from the Active Planes list. To create screen plane objects, select **Screen Plane**. (To create planar objects on the automatic working plane,

select **Automatic** as described in the next section.) The selected planar mode remains in effect until the next time it is changed.



The Active Planes list on the Shape tab of the Object Info palette switches the planar property of selected existing 2D objects. A selected 2D object drawn in screen plane mode can be switched to layer (planar), and vice versa.

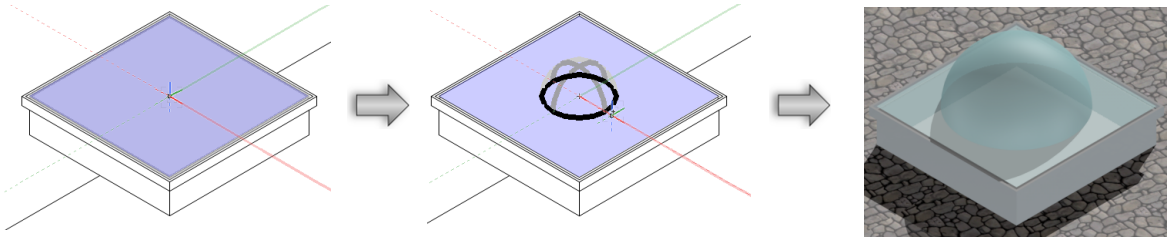


Click here for a video tip about this topic (Internet access required).

Depending on your workflow and preferences, the available options on the Active Planes list can be restricted, to never show the layer plane or never show the screen plane.

The Automatic Working Plane

In a 3D view, and when a typical drawing tool is selected, the automatic working plane is in effect. As the cursor moves over suitable drawing surfaces, the automatic working plane is highlighted on the surface to indicate that the drawing object can be placed there and will be drawn planar to the surface. “Automatic” displays in the Active Planes list.



With the Hemisphere tool selected, the automatic working plane displays on an applicable surface as the cursor moves over it. The axes display the working plane origin.

The hemisphere is created on the automatic working plane, and it is automatically made planar to the surface

Rendered view

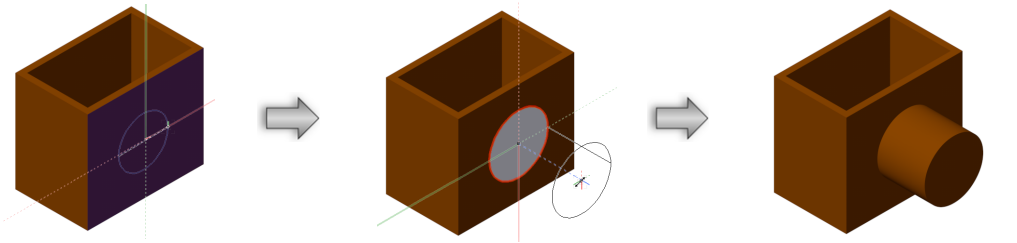
When the cursor is not over a suitable surface while in a 3D view, the default automatic plane is in effect. The default automatic working plane aligns to the layer plane, and an object drawn is placed on the layer plane. See “Understanding the Working Plane” on page 1166 for more information.

In some cases, it may be desirable to turn off the automatic working plane, as when drawing on the layer plane or screen plane, when drawing on a hidden surface where the automatic plane does not appear, or when drawing on a specific working plane that was established by the **Set Working Plane** tool. The \ key (backslash) toggles the automatic working plane on and off. This key can be customized in the Workspace Editor; see “Modifying Snapping and Mode Shortcuts” on page 1842. The color and opacity of the automatic working plane can also be customized; see “Configuring Interactive Display” on page 116.

Push/Pull Mode of Planar Tools

In a 3D view, and when the screen plane is not the active plane, planar objects such as circles, rectangles, rounded rectangles, ovals, arcs, polylines, and polygons can be extruded immediately after creation when the **Push/Pull** mode on the Tool bar is enabled for the tool. The mode’s toggle status applies to all these planar tools; if the mode is enabled

for one of the tools, it is enabled for all of them. This instantly and easily allows 3D objects to be created from planar objects, by moving the cursor or entering a distance in the data bar.



Use the Circle tool to create a circle on the side face of an object. The automatic working plane indicates a suitable planar surface.

The circle that was just created is still selected, and can immediately be extruded when the Push/Pull mode is enabled.

The extruded cylinder is created.

When the automatic working plane mode is enabled and the **Push/Pull** mode is in use over an object that supports solid operations, press and hold the Option key (Mac) or Alt key (Windows) and pull "out" to create a solid addition or push "in" to create a solid subtraction.

The **Push/Pull** mode of planar objects operates directly after creation of the object. If the object is deselected, it can be extruded later with the **Push/Pull** tool; see "Direct Modeling with the Push/Pull Tool" on page 357. In some cases, the **Push/Pull** mode may need to be toggled off, such as when creating several planar objects one after another.

Click here for a video tip about this topic (Internet access required).

Editing Objects
Understanding the Working Plane
The Active Planes List
Plane Mode Preferences

Drawing Structure

Organizing the Drawing

When you begin a drawing project, first develop a layer and class structure, along with a system for assigning items to the appropriate layers and classes. A layer is basically a container that holds items. Layers are comparable to the sheets of vellum that are used for hand-drafting; each item “belongs” to a layer in the same way that a hand-drafted item “belongs” to its vellum sheet. Additionally, in the Vectorworks Architect product, story levels and, optionally, design layers, can belong to building stories.

A class, however, is an attribute of an object. Classes span the layers and allow you to control the attributes and visibility of objects across multiple layers. To output specific elements of a drawing quickly and easily, simply select the appropriate layers and classes for display.

The program also provides ways to save the current drawing display with histories and saved views, and to present final drawings with viewports. Views and histories provide a “slice-of-time” look at the drawing. Histories record views of the drawing in a stack similar to the way Internet browsers do, while saved views store one or more aspects of the current display, such as the view, zoom, and layer and class visibility settings. Viewports are used to display final views of a drawing for client presentation.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Organization Dialog Box

Create and manage structural elements of the drawing using the Organization dialog box. This dialog box has specialized functions to sort and select its list items; see “List Box Functionality” on page 44.

To use the Organization dialog box:

1. Select **Tools > Organization**. Alternatively, click the **Classes** or **Layers** button on the View bar, or select **Edit View** from the **Saved Views** menu on the View bar.

   The Organization dialog box opens.

2. Select the appropriate tab for the element to be created or edited: Classes, Design Layers, Sheet Layers, Viewports, Saved Views, or References.

   In the Vectorworks Architect product, an additional Stories tab is available; see “Creating and Managing Stories” on page 174.

3. From the top of the dialog box, select either **Details** or **Visibilities** view.

   Generally, the Details view shows the current settings for each element, and enables editing of layer and class visibilities in the drawing area; the Visibilities view enables editing of layer and class visibilities in viewports and in saved views (see “Setting Visibilities in the Organization Dialog Box” on page 193).

4. Buttons at the bottom of each tab provide creation and management functions. Alternatively, right-click (Windows) or Ctrl-click (Mac) on a list item to display a context menu, which has most of the same functions as the tab buttons.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Classes Tab

If a description was entered for the class, it displays in a screen tip when you position the cursor over the class name. On Windows, click the disclosure arrow on the screen tip to collapse or expand it.

Click to show/hide the parameters.

Button	Function
New	Click **New** to open the New Class dialog box. See “Creating Classes” on page 177. When a new class is created, it does not automatically become the active class.
Edit	Select a class and click **Edit** to edit it in the Edit Class(es) dialog box. See “Setting Class Properties” on page 179. This option is not available when classes are displayed in hierarchical order and a class group header is selected.
Duplicate	Select a class and click Duplicate to create a copy of it. The name of the duplicate is the same as the original class, with a number added (as in cabinets-2); if the original name ends in a number, the next available sequential number is used. This option is not available when classes are displayed in hierarchical order and a class group header is selected.
Delete	Select a class and click **Delete** to open the Delete Class(es) dialog box. Specify what to do with the objects currently assigned to the class(es) being deleted (delete them, or reassign them to another selected class). Click **OK** to return to the Organization dialog box. Vectorworks moves all objects in the deleted class(es) to the appropriate class, or deletes them, as specified. This option is not available when classes are displayed in hierarchical order and a class group header is selected. Note that the Dimension and None classes cannot be deleted. These are default classes in every drawing.
Preview	Displays a preview of the current settings in the drawing area
Hierarchical display mode 	Click to toggle the hierarchical display of classes; additional options are available for controlling the hierarchical display at various levels, and for controlling visibility settings. See “Displaying Classes in Hierarchical Order” on page 159.

Design Layers Tab

The names of referenced design layers display in italics. If a description was entered for the design layer, it displays in a screen tip when you position the cursor over the layer name. If the layer is referenced, the screen tip also displays the full layer name and the source file name. On Windows, click the disclosure arrow on the screen tip to collapse or expand it.

Click to show/hide the parameters.

Button	Function
New	Click **New** to open the New Design Layer dialog box. See “Creating Layers” on page 162. When a new layer is created, it becomes the active layer.
Edit	Select a layer and click **Edit** to edit it in the Edit Design Layers dialog box. See “Setting Design Layer Properties” on page 165.

Button	Function
Duplicate	Select a design layer and click **Duplicate** to create a copy of it. The name of the duplicate is the same as the original layer, with a number added (as in floorplan-2); if the original name ends in a number, the next available sequential number is used.
Delete	Select a design layer and click **Delete**; when prompted, click **Yes** to confirm the deletion. When a design layer is removed from the drawing, all objects in that layer are also removed. Note that at least one design layer must remain in the drawing.
Preview	Displays a preview of the current settings in the drawing area
Update Reference (on the context menu only; layer import referencing method only)	This option is available if a design layer has been imported into this file with workgroup referencing. To update this file with layer information from the master file, right-click (Windows) or Ctrl-click (Mac) the layer, and select **Update Reference** from the context menu. See "Referencing Features" on page 207.
Level Types (Vectorworks Architect required)	Manages available level types; see "Managing Level Types" on page 174
Page Setup	Opens the Page Setup dialog box; see "Page Setup" on page 1761

Sheet Layers Tab

If a description was entered for the sheet layer, it displays in a screen tip when you position the cursor over the layer name. On Windows, click the disclosure arrow on the screen tip to collapse or expand it.

Click to show/hide the parameters.

Button	Function
New	Click **New** to open the New Sheet Layer dialog box. See "Creating Layers" on page 162. When a new layer is created, it becomes the active layer.
Edit	Select a sheet layer and click **Edit** to edit it in the Edit Sheet Layers dialog box. See "Setting Sheet Layer Properties" on page 170.
Duplicate	Select a sheet layer and click **Duplicate** to create a copy of it. The **Sheet Title** of the duplicate is the same as the original layer. The **Sheet Number** of the duplicate is the same as the original layer, with a number added (as in details-2); if the original **Sheet Number** ends in a number, the next available sequential number is used.
Delete	Select a sheet layer and click **Delete**; when prompted, click **Yes** to confirm the deletion
Preview	Displays a preview of the current settings in the drawing area

Viewports Tab

Click to show/hide the parameters.

Button	Function
New	Select **New** to open the Create Viewport dialog box. For sheet layer viewports, see "Creating a Sheet Layer Viewport from a Design Layer" on page 1616. For design layer viewports (Vectorworks Design Series required), see "Creating a Referenced Design Layer Viewport" on page 1622.

Button	Function
Edit	Select a viewport and then select **Edit** to edit it in the Properties dialog box. For sheet layer viewports, see “Viewport Properties” on page 1636. For design layer viewports (Vectorworks Design Series required), see “Viewport Properties” on page 1636.
Duplicate	Select a viewport and click **Duplicate** to create a copy of it. The name of the duplicate is the same as the original viewport, with a number added (as in details-2); if the original name ends in a number, the next available sequential number is used. The **Drawing Title** of the duplicate is the same as the original viewport. If the original viewport has a drawing label, the **Drawing Number** of the label in the duplicate viewport is the next available sequential number. Vectorworks places the duplicate viewport directly on top of the original, in the original sheet layer.
Delete	Select a viewport and click **Delete**; when prompted, click **Yes** to confirm the deletion
Preview	Displays a preview of the current settings in the drawing area

Saved Views Tab

Click to show/hide the parameters.

Button	Function
New	Click **New** to open the Save View dialog box; see “Creating Saved Views” on page 189
Edit	Select a saved view and click **Edit** to edit it in the Edit Saved View dialog box; see “Editing Saved Views” on page 190
Duplicate	Select a saved view and click **Duplicate** to create a copy of it. The name of the duplicate is the same as the original view, with a number added (as in deckview-2); if the original name ends in a number, the next available sequential number is used.
Delete	Select a saved view and click **Delete**; when prompted, click **Yes** to confirm the deletion

References Tab

Click to show/hide the parameters.

Button	Function
New	Click **New** to open the Open File dialog box. For layer import references, see “Adding and Editing Layer Import References” on page 210. For design layer viewport references (Vectorworks Design Series required), see “Creating a Referenced Design Layer Viewport” on page 1622.
Edit	Select a referenced file and click **Edit** to open the Edit Reference dialog box. For layer import references, see “Adding and Editing Layer Import References” on page 210. For design layer viewport references (Vectorworks Design Series required), see “Creating a Referenced Design Layer Viewport” on page 1622.
Delete	Select a referenced file and click **Delete**. In the Delete Reference dialog box, specify what to do with the items in the file that are currently referenced. See “Deleting References” on page 215.

Button	Function
Update	Select a referenced file and click **Update** to update this file with information from the master file. See “Updating References” on page 212.
Settings	Click **Settings** to open the Reference Settings dialog box; see “Setting the Referencing Options” on page 209

Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Organizing the Drawing
Layers
Classes
Saved Views
Workgroups and Referencing

Displaying Classes in Hierarchical Order

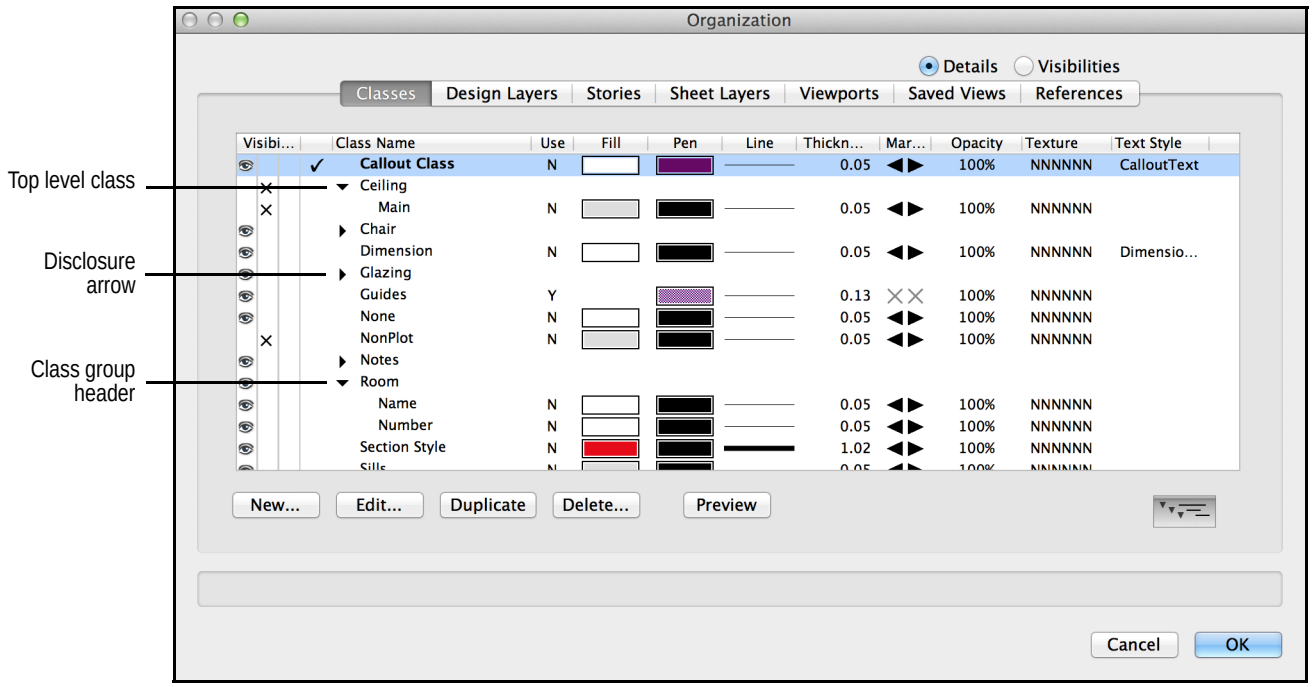
On pop-up menus (including in the Object Info palette, the View bar, and dialog boxes), in the Organization dialog box, and (for Vectorworks Design Series products) in the Navigation palette classes with compound names separated by a dash, such as Walls-Ext, can be displayed in hierarchical order (up to four levels). Each level of a class is assigned a group header at the top level when displayed in hierarchical order. A number of options are available to control the hierarchical display of classes at various levels.

To enable/disable hierarchical order display on pop-up menus throughout Vectorworks software:

- On the Session tab of the Vectorworks Preferences dialog box, select/deselect **Display classes in pop-up menus hierarchically** to toggle between hierarchical and non-hierarchical display. This setting controls all pop-up menus throughout the software.

To enable/disable hierarchical order display in the Organization dialog box and the Navigation palette:

- In the Organization dialog box Classes tab, click the toggle button in the lower right corner or select the **Hierarchical Display** context menu option to toggle between hierarchical and non-hierarchical display; whatever expand/collapse state all items were last in is displayed
- In the Navigation palette (Vectorworks Design Series required), select the **Hierarchical Display** context menu option to toggle between hierarchical and non-hierarchical display; whatever expand/collapse state all items were last in is displayed
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



When classes are displayed in hierarchical order, the list's sort key and sort order cannot be changed.

To expand/collapse a single class group header in hierarchical order display:

- Click the disclosure arrow to the left of a class group header
- Double-click the class group header

To expand all classes in the file or to collapse the list to only top-level classes (those with no dash in the name) and class group headers in hierarchical order display:

- Shift+Option-click (Mac) or Shift+Alt-click (Windows) the disclosure arrow to the left of a class group header
- Shift+Option-double-click (Mac) or Shift+Alt-double-click (Windows) on the class group header

Alternatively, select **Expand All** or **Collapse All** from the context menu.

To expand/collapse all class subgroups in hierarchical order display:

- Option-click (Mac) or Alt-click (Windows) the disclosure arrow to the left of a class group header
- Option-double-click (Mac) or Alt-double-click (Windows) on the class group header

To manage visibility settings in hierarchical order display:

- If all class sublevels contain the same visibility setting, then their class group header is also automatically assigned that same visibility setting; otherwise, the **Visibility** column is blank for the class group header
- Assign or change a visibility setting in the **Visibility** column for the class group header, and all of its sublevels inherit that visibility setting

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Organization Dialog Box
Classes
List Box Functionality
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing Drawing Structure from Standards or Other Files

Specific aspects of the drawing structure (classes, sheet layers, and design layers) can be imported from one or more existing files. The files from which you import can be the standard files provided with the Vectorworks software, or other files that you have created.

The standard files (.sta files) are located in the Standards folder (in [Vectorworks]/Libraries/Defaults). Layer standards use only approved layer names for a particular industry. Class standards use approved name and class settings for a particular industry. Custom layers or classes can be created in a blank file and saved as an .sta file in the Standards folder. Once they are placed in the Standards folder, these new layers or classes become part of the Standards list and are available when new layers or classes are created.

If one of your work files becomes corrupted, a possible recovery method is to import the data into a new file. Create a new file and then import classes, sheet layers, and design layers (and optionally, design layer objects) from the original file as needed.

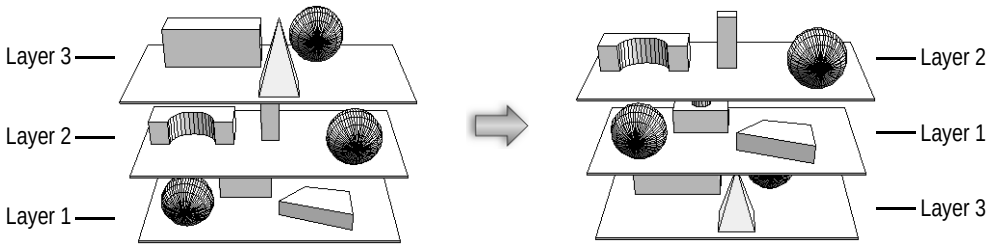
To import drawing structure, use the Organization dialog box to create a new class, sheet layer, or design layer. Instead of creating a new item, however, select the import option. Select the file from which to import, and then select the specific classes or layers from that file to import. For a design layer, select whether to import the objects on the design layer, as well as the layer itself.

See “Creating Layers” on page 162 and “Creating Classes” on page 177 for details.

Click here for a video tip about this topic (Internet access required).

Layers

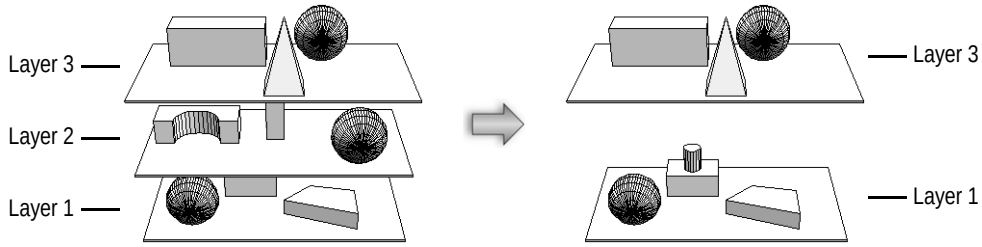
Create layers in a drawing to provide immediate flexibility. Stack the design layers in any order, or temporarily hide some of them. Reorder the layers, which moves all of the objects contained within each layer to another location without actually modifying the objects or their alignment with each other.



Layer order rearranged

If you hand-draw a floor plan on two design layers, one vellum sheet with a master drafting plan and another with an addition to the plan, it would be easy to look at the floor plan with or without the addition. In the Vectorworks program, the vellum is electronic, so far more can be done with it. A distance can be set between design layers rather than having them lie flat on top of each other. Additionally, with the program’s modeling capabilities, these layers can be viewed in 3D. For example, if the first floor, second floor, basement, and roof of a house are each placed in their own design layers, not only can the 2D drafting plan be printed for any one of those layers, but the design layers can be linked together, creating a model of a fully formed 3D house. Use viewports to display several views of the finished design, either on design layers (Vectorworks Design Series required) or on sheet layers, which are special presentation layers. The original design layers remain unchanged.

Layers have many other uses, as well. Move elements between design layers, or change the scale of a layer, instantly making a detail of an area of the drawing without re-drawing anything. Create design layers with objects that should always display, or layers that contain objects for display only at particular times. Control the visibility of the design layers to limit the need for creating new objects.



In the Vectorworks Architect program, design layers can optionally be associated with stories that contain objects; stories define absolute elevations in the building model, while layers can be set at an elevation relative to the story. This method of organizing a file makes it much easier to manage a building's layers and certain associated objects like walls and columns. See "Setting Up the Building Structure with Stories" on page 172.

Use design layers to draw and model projects. Use sheet layers to create a presentation version of the finalized drawing; this can include viewports, title blocks, notes, and other annotations (see "Creating Sheet Layer Viewports" on page 1616).

On the layers list in the View bar, sheet layers are listed first, and then design layers. A separator divides the two types of layers in the list.

Sheet layers display with a wide gray border representing the print margin area, as opposed to design layers, which have a thin gray border (when the page boundary is displayed). This makes it easier to distinguish the layer types at a glance.

Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Layers

When a new drawing is created, it automatically contains an empty design layer entitled "Design Layer-1." Add design layers to the drawing as needed to organize it. Add sheet layers as needed for presentation. Create new design layers and sheet layers, or import them (and optionally, the objects they contain) from other current version files or from standard files. In Vectorworks Design Series products, create a design layer viewport to reference design layers in other files without importing them.

 To create a new layer:

1. For convenience, a new layer can be created from multiple locations in the Vectorworks software.
   - **New** button on the Design Layers/Sheet Layers tab of the Organization dialog box (**Tools > Organization**)
   - **Layers** button on the View bar to open the Organization dialog box
   - **New Design Layer** or **New Sheet Layer** option from the Layers drop-down list on the View bar
   - **Layer** field on the Shape tab of the Object Info palette
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- **New** context menu on the Design Layers or Sheet Layers tab of the Navigation palette (Vectorworks Design Series required)

2. From the New Design Layer or New Sheet Layer dialog box, create a new layer, or import a layer and its properties from standard or existing Vectorworks files.

Click to show/hide the parameters.

Parameter	Description
Create a New Design Layer or Create New Sheet Layer	Creates a new design or sheet layer. • For a design layer, also enter a descriptive **Name**. • For a sheet layer, enter a unique **Sheet Number**, or use the default (as in Sht-2). The **Sheet Number** can be displayed in drawing labels, section markers, and title blocks on sheet layers. Also enter a descriptive **Sheet Title**, which can be displayed in title blocks.
Import Design Layers or Import Sheet Layer	Imports layers and their attributes from standard files or from existing files. Files located in the Standards folder, as well as existing files selected previously, are displayed in the list. Select a file; the available layers and descriptions are listed beneath the file name. If a layer name in the current file matches a layer in the import file, that layer is not on the list as an import option. Select the desired layer(s). To select multiple layers from the import list, hold down the Ctrl (Windows) or Command (Mac) key while you click.
Choose	Click **Choose** to select a file for layer import. Files must be in the current version.
Import Layer Objects (design layer only)	In addition to importing the design layer structure and attributes, imports the contents of the layers; if a file has become corrupted, this is a possible way of recovering its data. If there is a naming conflict with pre-existing symbols or pre-existing layers in the current document, replace or rename the symbols or rename the layers.
Creation Options	
Saved View Visibility (design layer only)	Sets the visibility of the new design layer in saved views (when saved views exist in the drawing)
Viewport Visibility (design layer only)	Sets the visibility of the new design layer in viewports (when viewports exist in the drawing)
Edit Properties After Creation	Immediately after creation, opens the Edit Design Layers or Edit Sheet Layers dialog box to set the properties of the new layer(s)

3. Click **OK** to create the new design or sheet layer(s).

   The layers display in the Layers list on the Organization dialog box, View bar, Object Info palette and (for Vectorworks Design Series) the Navigation palette. If the layer was created from anywhere other than the Object Info palette, it becomes the active layer.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Design Layer Properties
Setting Sheet Layer Properties
Assigning Objects to Classes and Layers
Setting Class and Design Layer Options
Setting Visibilities in the Organization Dialog Box
The Visibility Tool
Layers
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Active Layer

There are several ways to change the active design layer or sheet layer.

To be able to add, remove, or edit objects on a design layer, either the layer must be active or the layer options must be set to allow modifications to other layers (see "Setting Class and Design Layer Options" on page 193).

Setting the Active Layer in the Organization Dialog Box

To set the active layer:

1. From the Organization dialog box, in **Details** view, select the Design Layers tab or the Sheet Layers tab.

   The active layer is indicated by a check mark to the left of the **Design Layer Name** or **Sheet Number**. The layer also is highlighted in bold text.

2. To make a different layer active, click the column to the left of its name/number.
3. Click **OK**.

   The dialog box closes and the active layer displays.

Setting the Active Layer in the View Bar

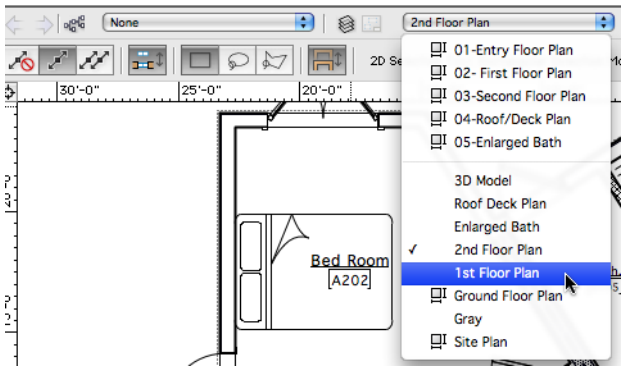
To set the active layer:

1. Click the Layers list on the View bar to display a list of all of the sheet layers (top section) and design layers (bottom section) in the drawing.

   On Mac, the active layer is indicated by a check mark; on Windows, the layer is highlighted in bold text.

2. Click the layer to be activated.

   The layers list closes and the active layer displays.



Setting the Active Layer in the Navigation Palette

To set the active layer:

1. From the Navigation palette, select the Design Layers tab or the Sheet Layers tab.

   The active layer is indicated by a check mark to the left of the **Design Layer Name** or **Sheet Number**. The layer also is highlighted in bold text.

2. To make a different layer active, click the column to the left of its name/number.

   Alternatively, Right-click (Windows) or Ctrl-click (Mac) on the layer to be activated and select **Activate** from the context menu.

Setting the Active Layer Using the Shortcut Keys

If there is a small number of layers, switch among layers with the **Switch active layer/class** shortcut key combination specified in Vectorworks preferences (see “Edit Preferences” on page 49). This selects a layer by moving up or down through the layer list one layer at a time.

Setting the Active Design Layer in the Document Context Menu

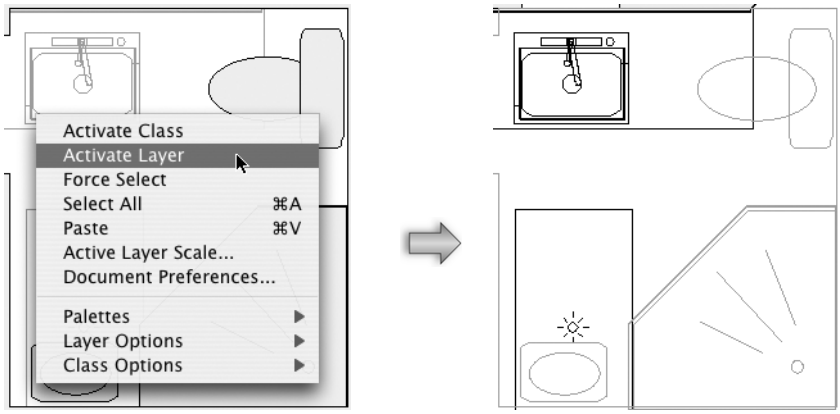
If multiple design layers are set to be visible, and the layer options are set to show those other layers, the drawing area may display objects that are on non-active layers. Use the **Activate Layer** command to make the layer of one of these objects active.

The **Force Select** command on the document context menu also changes the active class and layer (if necessary), and selects the clicked object.

To set the active design layer:

1. In the drawing area, right-click (Windows) or Ctrl-click (Mac) a visible object on a non-active design layer.
2. From the document context menu, select **Activate Layer**.

   The object’s layer becomes active.



Right-click (Windows) or Ctrl-click (Mac) on the sink from a gray, non-active layer to open the document context menu

The Activate Layer command activates the sink’s layer, and the objects that are not in that layer become grayed

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Design Layer Properties
Setting Sheet Layer Properties

Setting Design Layer Properties

Once created, the design layers display on the Design Layers tab of the Organization dialog box, where various properties can be set and edited.

To edit design layers:

1. Select **Tools > Organization**. Alternatively, click the **Layers** button on the View bar.

   The Organization dialog box opens.
2. Select the Design Layers tab.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

A list of the current layers in the drawing displays in their stacking order. Depending on which view option is selected at the top of the dialog box, either details or visibilities of the design layers display. Stacking order, drawing area visibility, and active layer can be changed in **Details** view. The visibility of layers in viewports and in saved views can be changed in **Visibilities** view.

In the Vectorworks Architect product, **Details** view includes the stories and story levels for each layer that is associated with a story level.

Layers that are imported from another file with layer import referencing display in italics. Position the cursor over the layer name to display a screen tip with the full layer name and the source file name.

3. To change other layer properties, select one or more layers and click **Edit** to open the Edit Design Layers dialog box.

Click to show/hide the parameters.

Parameter	Description
Name	If one layer was selected, displays the layer's name; edit as needed
Description	Click to enter additional information about the design layer. The description displays as a screen tip when you position the cursor over the design layer name in the Organization dialog box, Navigation palette, or (on Mac only) the Layers list on the View bar. Descriptions also display in the New Design Layer dialog box, so that you can see descriptions of layers before you import them.
Scale	Sets the scale for the selected layer(s); see "Changing the Scale of Selected Design Layers" on page 68
Stacking Order	Changes the 2D stacking order of the layer(s); see "Changing the Design Layer Stacking Order" on page 167
Story (Vectorworks Architect required)	Associates the layer with a story; select the story from the list, or create a new story for the layer (see "Creating and Managing Stories" on page 174).Layers that are associated with story levels derive their elevation value from the associated level. If the design layer does not need to belong to a story and one of its levels, select None; for example, a detail layer does not need to be associated with a building story.
Story Level (Vectorworks Architect required)	Associates the layer with a story level; select the level from the list of levels for the story selected in **Story**. The level elevations are displayed in brackets to assist with choosing the correct level. Alternatively, create a new level (see "Default Story Levels" on page 173).
Elevation	Sets the height of a layer relative to the ground plane. For Vectorworks Architect users, the elevation is set relative to the story, if the layer is associated with a story level, and the elevation cannot be changed here. The elevation relative to the ground plane also displays. For example, the second story of a building might be slightly offset from the story elevation, and also several feet above street level. When a new layer is added to a drawing, its elevation values are automatically entered based on the previous layer's elevation and layer wall height values.
Layer Wall Height	Indicates the height of walls (and certain other objects such as columns) created on that layer, when the **Top Bound** option for those objects is set to Layer Wall Height
Opacity	Controls how the selected layer(s) and other visible layers display; on Windows, the Vectorworks display preference for GDI+ imaging must be enabled

Parameter	Description
Transfer Mode (Windows only)	Controls how the selected layer(s) and other visible layers display; the Vectorworks display preference for GDI+ imaging must be disabled
Renderworks Background	When the Renderworks product is installed, select the Renderworks background to use for the selected layer(s) from either the default content or the current file's content. If the Renderworks Background choices are not available, the background selection is controlled by the Renderworks style in effect. Set the background in the Renderworks style instead; see "Renderworks Styles" on page 1596.
Colors	Specifies the default layer color for the selected layer(s)
Saved Views	Specifies the visibility settings for the selected layer(s) in each saved view; see "Setting Visibilities in the Organization Dialog Box" on page 193
Viewports	Specifies the visibility settings for the selected layer(s) in each viewport; see "Setting Visibilities in the Organization Dialog Box" on page 193
Georeferenced (Vectorworks Architect or Landmark required)	Enables georeference information to be attached to the layer(s); click **Edit Georeferencing** to open the Georeferencing dialog box. See "GIS and Georeferencing" on page 775 for details.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

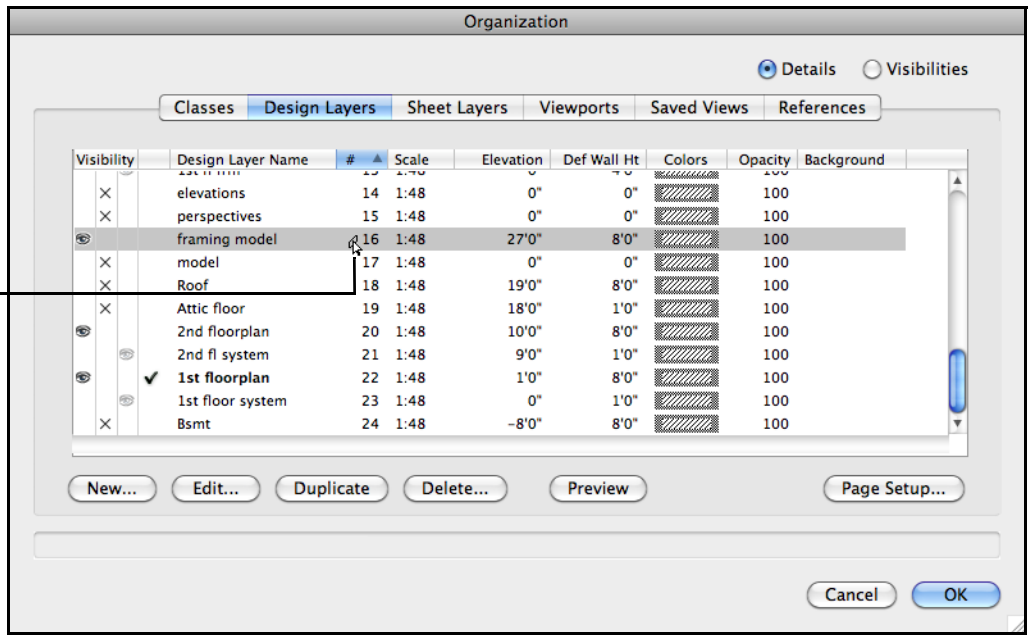
Setting the Design Layer Opacity
Setting the Design Layer Transfer Mode
Creating Layer Backgrounds
Setting the Design Layer Color
Setting Up the Building Structure with Stories

Changing the Design Layer Stacking Order

Design layers are viewed and printed in "stacking order," the top-to-bottom order in the Layers Setup dialog box. Initially, layers are "stacked" in the order in which they are created, but their order can be changed at any time.

To change the stacking order of design layers:

1. From the Organization dialog box, select the Design Layers tab in **Details** view to see the current stacking order. The # column to the right of the layer name indicates the layer's current position in the stack, with 1 being the top layer.
2. Use one of the following methods to change the layer stacking order:
   - The # column must be the current sorting column to change the stacking order; if the list is not currently sorted by stacking order, click the # column heading to change it. Then click the # column of the layer(s) to be moved, and drag it up or down the list. A horizontal line indicates where the layer(s) will be inserted in the current order.
   - Select the layer(s) to move, and then click **Edit** (or double-click a layer row) to open the Edit Design Layers dialog box. In the **Stacking Order** field, enter the number for the new stacking order position of the layer(s). Click **OK** to accept the changes.
3. The Design Layers tab displays the new stacking order. Click **OK** to close the Organization dialog box and save the changes.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click on the layer's # column and drag it up or down the list to change the layer's stacking order

Setting the Design Layer Opacity

This feature is available on all Mac systems, and on Windows systems only when the GDI+ imaging preference is enabled. (See "Vectorworks Display Preferences" on page 50 for information about setting this preference.) Design layers have a setting that controls the transparency of layer objects that overlap visible objects in another layer. An **Opacity** value of 100% means that nothing beneath the active layer is visible. Decrease the **Opacity** value to increase the transparency of the objects on the layer.

To set the opacity for a design layer:

1. From the Organization dialog box, select the Design Layers tab. Select one or more layers, and then click **Edit**.

   The Edit Design Layers dialog box opens.

2. Drag the **Opacity** slider to the left to increase the transparency, or enter an opacity percentage (0-100) in the box to the right of the slider.

3. Click **OK** to close the Edit Design Layers dialog box, and then click **OK** again to close the Organization dialog box.

   Individual objects can also have an opacity percentage applied. See "Opacity Attributes" on page 1101.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Design Layer Transfer Mode

Setting the Design Layer Transfer Mode

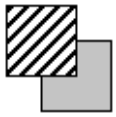
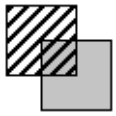
This feature is only available on Windows systems. When the GDI+ imaging preference is disabled, design layers have a setting that controls the display of layer objects that overlap visible objects in another layer. (See "Vectorworks Display Preferences" on page 50 for information about setting this preference.)

To set the transfer mode for a design layer:

1. From the Organization dialog box, select the Design Layers tab. Select one or several layers, and then click **Edit**.

   The Edit Design Layers dialog box opens.

2. Select the desired **Transfer Mode** from the list. Click **OK** to close the Edit Design Layers dialog box, and then click **OK** again to close the Organization dialog box.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mode	Description
Paint	Makes objects in the new layer solid, obscuring objects in layers stacked below it (this is the default setting) 
Overlay	Makes it so objects in the new layer do not obscure stacked layers 
Invert	Makes a reversed, or photo-negative image display when an object in the new layer overlaps an object in another layer 
Erase	Makes objects in the new layer display all foreground patterns as white and all background patterns as transparent 
Not Paint	Makes objects in the new layer solid and inverts any areas that overlap objects in stacked layers 
Not Overlay	Makes objects in the new layer transparent and inverts layer colors 
Not Invert	Makes objects in the new layer transparent and converts any black pixels from overlapping areas to white and white pixels to transparent 
Not Erase	Makes objects in the new layer transparent and converts any white pixels from overlapping areas to black and black pixels to transparent 

Most printer devices do not support all of these modes, especially PostScript printers and vector devices such as pen plotters. The **Rasterize print output** option may produce the best results for certain transfer modes. The use of color in transfer modes may produce color blending.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Design Layer Opacity
Printing a File

Setting the Design Layer Color

The fill and pen color of objects that are drawn on or moved to a design layer can be controlled by the color settings of the design layer. The **Use layer colors** setting in the Document Preferences dialog box must be turned on (see "Document Display Preferences" on page 60).

These settings are overridden by the **Black and white only** option in the Document Preferences dialog box, even with the **Use layer colors** option selected.

To control the color of objects by their design layers:

1. From the Organization dialog box, select the Design Layers tab. Select one or several layers, and then click **Edit**.

   The Edit Design Layers dialog box opens.

2. Click **Colors**.

   The Color Defaults for Layer dialog box opens.

3. For both the fill and pen, set the **Foreground** and **Background** colors by clicking the appropriate list and selecting a color from the main Color Menu dialog box. A preview example is shown at the bottom of the dialog box.

   The fill background color controls the appearance of objects with a solid fill.

4. Click **OK**.

   When the **Use layer colors** preference is selected, all objects on the layer are drawn with the specified colors.

   Viewports have separate control of layer color (see "Advanced Sheet Layer Viewport Properties" on page 1642).

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Colors

Setting Sheet Layer Properties

Once created, the sheet layers display on the Sheet Layers tab of the Organization dialog box, where various layer properties can be set and edited.

Sheet layers are always at a 1:1 scale, Active Only, and set to Top/Plan view.

 To edit sheet layers:

1. Select **Tools > Organization**. Alternatively, click the **Layers** button on the View bar.

   The Organization dialog box opens.

2. Select the Sheet Layers tab and the **Details** view.

   The Sheet Layers tab opens, with a list of the current layers in the drawing. The layer stacking order and the active layer can be changed in **Details** view. **Visibilities** view does not apply to sheet layers.

3. To change layer properties, select one or more layers and click **Edit** to open the Edit Sheet Layers dialog box.

Click to show/hide the parameters.

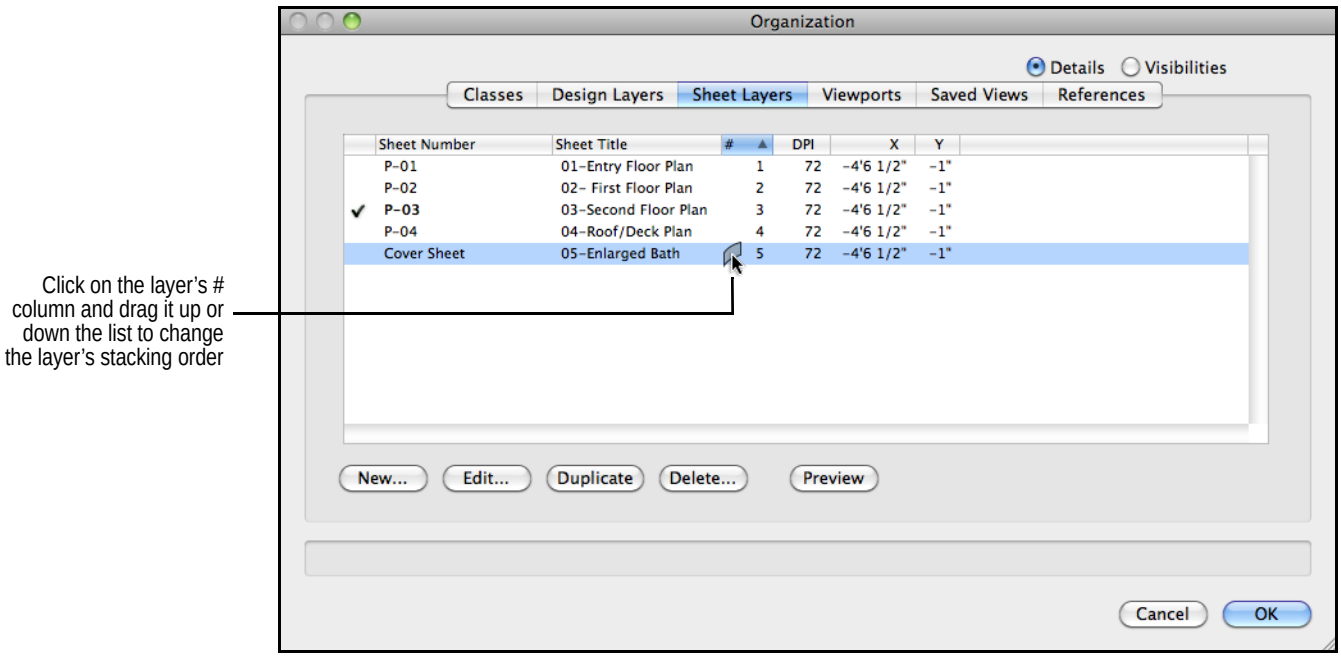
Parameter	Description
Sheet Number	Displays the number of the selected sheet layer; this number must be unique in the document. If this value is changed and **Use Automatic Drawing Coordination** is enabled in document preferences (Vectorworks Design Series required), any annotation objects (sheet borders, drawing labels, or section markers) that show this number are updated automatically.
Description	Click to enter additional information about the sheet layer. The description displays as a screen tip when you position the cursor over the sheet number in the Organization dialog box, Navigation palette, or (on Mac only) the Layers list on the View bar. Descriptions also display in the New Sheet Layer dialog box, so that you can see descriptions of layers before you import them.
Sheet Title	Displays the descriptive title of the selected sheet layer. If **Use Automatic Drawing Coordination** is enabled in document preferences (Vectorworks Design Series required), any sheet borders that show this title are updated automatically.
Stacking Order	Changes the 2D stacking order of the layer(s); see “Changing the Sheet Layer Stacking Order” on page 171
Raster Rendering DPI	Specifies the resolution (Dots Per Inch) for printing and viewing the raster rendered viewports in the selected sheet layer (does not affect vector geometry or bitmaps that are inside viewports); see “Setting the Print Resolution” on page 1765 for more details
Origin	Specifies the X and Y coordinates of the sheet layer origin; each sheet layer has its own origin
Page Setup	Opens the Page Setup dialog box for entry of sheet layer printing parameters; this saves time later, when different sheet layers can be automatically sent to different printers with different print area settings (see “Page Setup” on page 1761)

Changing the Sheet Layer Stacking Order

Sheet layers are viewed and printed in “stacking order,” the top-to-bottom order in the Organization dialog box. Initially, layers are “stacked” in the order in which they are created, but their order can be changed at any time.

To change the stacking order of sheet layers:

1. From the Organization dialog box, select the Sheet Layers tab in **Details** view to see the current stacking order. The # column to the right of the sheet title indicates the layer’s current position in the stack, with 1 being the top layer.
2. Use one of the following methods to change the layer stacking order:
   - The # column must be the current sorting column to change the stacking order; if the list is not currently sorted by stacking order, click the # column heading to change it. Then click the # column of the layer(s) to be moved, and drag it up or down the list. A horizontal line indicates where the layer(s) will be inserted in the current order.
   - Select the layer(s) to move, and then click **Edit** (or double-click a layer row) to open the Edit Sheet Layers dialog box. In the **Stacking Order** field, enter the number for the new stacking order position of the layer(s). Click **OK** to accept the changes.
3. The Sheet Layers tab displays the new stacking order. Click **OK** to close the Organization dialog box and save the changes.



Setting Up the Building Structure with Stories

The Vectorworks Architect product includes an additional document structural feature called stories. Stories define absolute elevations for the various floors of a building, and allow architects to manage defined story levels within those floors for construction elements like slabs, finish floors, and ceilings. The levels in a story have a Z height relative to the story; the story controls their absolute elevation, and if the story elevation changes, the levels and certain objects that are associated with them automatically change their elevation or boundaries along with the story.

Story levels can be optionally associated with a layer that contains objects (such as furniture, fixtures, and walls). To add objects to a story, at least one layer is required. The story levels that make up a story act as a top or bottom constraint for special "bounded" objects. These bounded objects can be defined at their boundaries by the levels that make up a story, such as a slab or a ceiling. For example, an exterior wall (or one of its components) can extend from the top of the slab on one story to the top of a slab on the story above it. Slabs, walls, wall components, curtain walls, spaces, stairs, escalators, columns, and pilasters can be set according to a top and bottom boundary layer; the location of these boundaries is controlled by the elevation of the level and ultimately by the story elevation. Both walls and slabs can have their dynamic height information set and saved in a wall or slab style, so they can automatically take on their defined height condition. This method of using bounded objects allows flexibility and accuracy when defining a model, from the early to the final stages of design.

Begin project setup by creating the stories and their associated levels. Stories are created and managed from the Organization dialog box; either pre-defined standards or custom standards can be used. As the stories are added, any associated design layers are also created according to story level definitions. Finally, associate the applicable objects to the story level at the object's top and bottom boundaries, creating a cohesive model that is structured and controlled by its building stories.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Default Story Levels
Creating and Managing Stories
Organizing the Drawing
The Organization Dialog Box
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Default Story Levels

Stories contain levels with names and parameters defined by standards. When a story has a level, it can be used as a potential boundary for objects in its story (such as walls, slabs, and so on) or for the story above or below it. You can select the automatically defined levels when creating a story, or create customized levels by defining the default story levels before creating the stories. Certain projects may require custom level types, which can also be set up in advance.

To create or edit default story levels:

1. Select **Tools > Organization** to open the Organization dialog box. Click the Stories tab.

   The functionality of the Organization dialog box is described in "The Organization Dialog Box" on page 155.

2. Click **Default Story Levels**.

   The Default Story Levels dialog box opens, listing the currently defined default levels, their elevations (offset from the story elevation), and any associated layer names.

3. Click **New** to create a new default story level, or **Edit** to modify the currently selected default level.

   The New Default Story Level or Edit Default Story level dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Level Type	Select the type of level from the list of standard levels, or click **New Level Type** to add the name of a custom level type to the list
Elevation	Sets the elevation of the layer, relative to the story.
Create Layer	Creates a design layer, and associates the level with the layer; this is necessary when the level and associated layer contains objects such as furniture, fixtures, walls, windows, and so on. When a level is only being used as an elevation reference for bounded objects, an associated layer is not needed. When an associated layer is no longer required and **Create Layer** is deselected, an alert displays. Select **Yes** to delete the layer, or **No** to keep the layer in the file but disassociate it from the level.
Name	Specifies the name of the associated layer. This name is used with a story-specific prefix or suffix to create the actual layer name.
Scale	Click **Scale** to set the scale to use for the associated layer; see "Design Layer Scale" on page 67
Layer Wall Height	Sets the default height for walls and columns that belong to the associated layer; the object's settings must define its top height by the layer wall height for the object to use this height

4. By default, several level types are available, such as slab, finish floor, and ceiling; default story level types are provided as default content (see "Resource Libraries" on page 219). To create a custom level type, select New Level Type from the **Level Type** list.

   The New Level Type dialog box opens. Enter the name of the new type of level.

5. Click **OK** to return to the New/Edit Default Story Level dialog box.
6. Once the default story level has been defined or modified, click **OK** to return to the Default Story Levels dialog box.
7. Once the default story levels in the list have been defined or modified, click **OK** to return to the Organization dialog box.

Stories can now be created. They will use the default information specified to create the levels associated with the stories.

Managing Level Types

Available story levels can be managed from the Organization dialog box.

To manage available level types:

1. Select **Tools > Organization** to open the Organization dialog box. Click the Design Layers tab.
2. Click **Level Types**.

   The Level Types dialog box opens. Specify the level types that are available by default when creating stories.

Click to show/hide the parameters.

Parameter	Description
Level Type Name	Lists the current default level types
New	Opens the New Level Type dialog box, to create a new level type
Edit	Opens the Edit Level Type dialog box, to edit the name of the level type
Delete	Deletes the currently selected level type

3. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating and Managing Stories
The Organization Dialog Box
Standard Viewports
Standard Naming

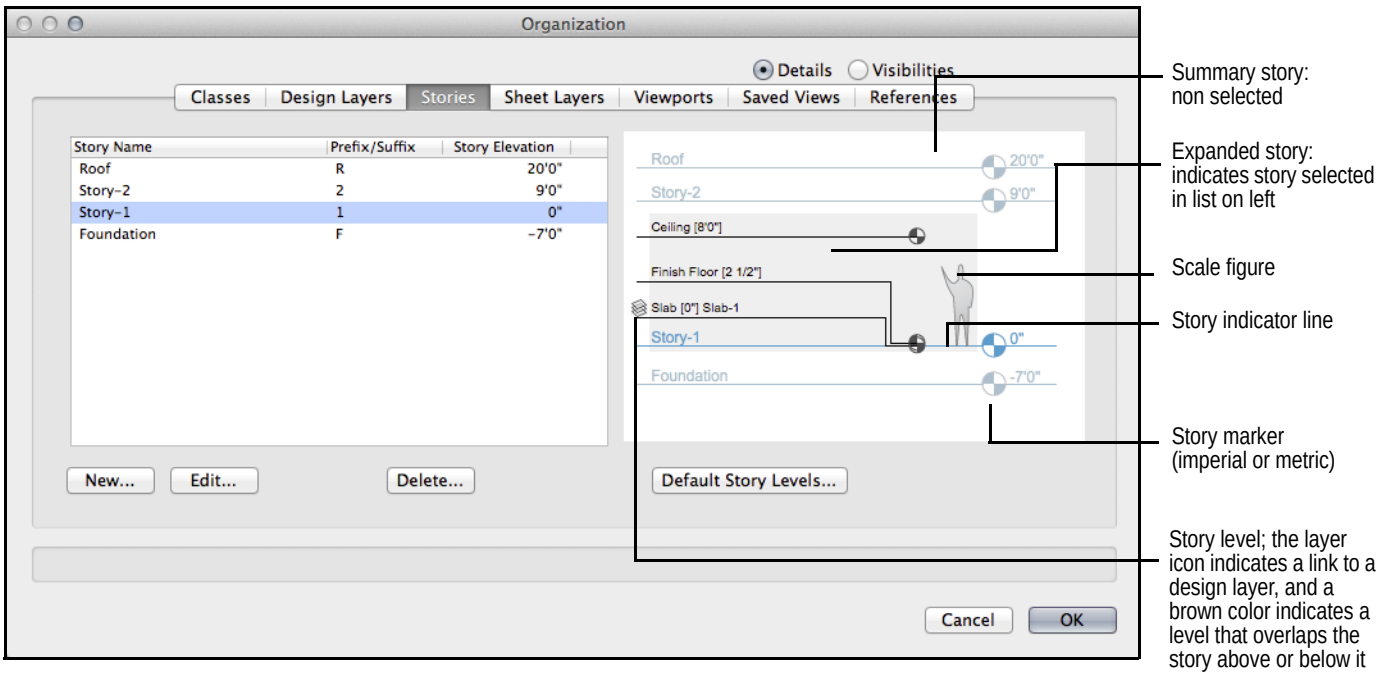
A Creating and Managing Stories

Project setup begins with creating stories and specifying their associated story levels. The Stories tab of the Organization dialog box allows stories to be created and managed. Stories can also be created when creating or editing design layers with options that display only when the Vectorworks Architect product is installed. See “Setting Design Layer Properties” on page 165.

To create and manage stories from the Organization dialog box:

1. Select **Tools > Organization** to open the Organization dialog box. Click the Stories tab.

   The functionality of the Organization dialog box is described in “The Organization Dialog Box” on page 155.
2. Visibilities view displays a list of story names. From the top of the dialog box, select **Details** view.
3. On the left, the stories are listed, along with the story’s prefix or suffix, and elevation. On the right, an interactive diagram displays the stories and associated layers that make up the building model.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click to show/hide the parameters.

Parameter	Description
Story Name list	Lists the stories in the building model, along with each story's prefix or suffix designation and elevation. The story order cannot be changed; stories are always listed in elevation order.
New	Creates a new story and specifies the associated levels, prefix or suffix, layer (if any), and elevation
Edit	Edits the selected story's name, suffix, or elevation
Delete	Deletes the selected story or stories; when prompted, select whether to delete all layers associated with the story or stories
Default Story Levels	Opens the Default Story Levels dialog box, for defining or editing the defined layer defaults; see "Default Story Levels" on page 173

4. Click **New** to create a new story, or **Edit** to modify the currently selected story.

   The New Story or Edit Story dialog box opens. Specify the name, elevation, prefix or suffix for any layers to be created, and associated levels. To add objects to a story, at least one layer is required.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies the name of the story
Story Elevation	Sets the story elevation; a default value is suggested, based on the elevation of other stories
Layer Name Suffix/Prefix	When design layers are associated with story levels, specifies the prefix or suffix to append to associated design layer names, to distinguish the layers from other design layers. The design layer name can be changed later if needed.

Parameter	Description
Level List	Lists the available levels to associate with the story; select a level to include by clicking in the first column. Associated levels are indicated with a check mark. Only one level of a given type can be associated with a story; the available levels are determined by the default story levels. When a story layer includes a level, it can be used as a potential boundary for objects on layers in its story, or in the story above or below it. Selecting level types for the story is optional; however, level types are necessary when setting up a model with bounded objects.
New Level	Opens the New Story Level dialog box, to add a new level type
Edit Level	Opens the Edit Story Level dialog box, to edit the selected level type
Edit Default Story Levels	Opens the Default Story Levels dialog box, for editing or adding to the levels available in the story; see “Default Story Levels” on page 173

5. If changing the elevation of a story from the Edit Story dialog box, the Change Story Elevation dialog box opens to determine how to adjust the story and the stories around it. If not changing the story elevation, proceed to Step 7.

Click to show/hide the parameters.

Parameter	Description
Move this Story and all Stories above it	Changes the elevation of the story, and adjusts all stories above it, including associated levels, layers, and layer-defined objects
Move this Story only	Changes the elevation of the current story and its associated levels, layers, and objects only; other stories remain at their set elevation
Move this Story and all Stories below it	Changes the elevation of the story, and adjusts all stories below it, including associated levels, layers, and layer-defined objects
Move all Stories	Changes the elevation of the story, and adjusts all stories above it and below it, including associated levels, layers, and layer-defined objects

6. Click **OK** to return to the Edit Story dialog box.
7. When the story settings have been made, click **OK** to return to the Organization dialog box.

   The stories, once set up with associated levels and any layers with layer-defined objects, define the building model. On the Design Layers tab of the Organization dialog box, listed layers that are associated with levels display their story, level type, elevation, and default wall height.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Organizing the Drawing
Default Story Levels
Standard Viewports
Standard Naming

Classes

In addition to design layers, classes are a powerful way to organize the elements in a drawing project according to category. This allows the objects to be viewed, changed, and tracked as a group. Because classes work across design layers, they allow the grouping of similar objects in a drawing that for practical reasons need to exist on separate layers.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Classes also allow the same file to be used for all stages of a project and for various purposes. For example, the classes shown for a license application could be different from those shown for the building contractor.

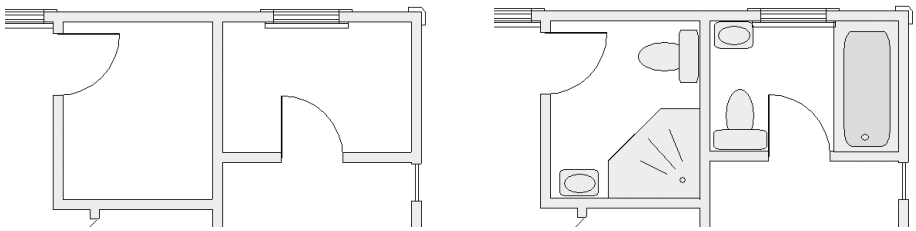
> Vectorworks classes are similar in function to—and are exported as—AutoCAD layers. If a drawing will be exported to AutoCAD, use classes to make it easy to turn on or off selected portions of the drawing. For example, if a consultant using AutoCAD will be doing the duct layout for a building, a furniture class allows him or her to turn off the furniture layer, instead of deleting furniture objects.

Setting up the classes at the beginning of a project is recommended, so that objects can be assigned to appropriate classes as they are created.

Each new drawing created with the Vectorworks program automatically has two classes: Dimension and None. Any dimensions created are assigned, by default, to the Dimension class (this is a preference setting that can be changed; see “Dimension Preferences” on page 61). Group objects are assigned to the active class. All other objects and symbols are assigned to the None class, which is the default active class. These two classes can be renamed but not deleted.

If the drawing was created from a template, other classes may have been provided. New classes can be created, duplicated, edited, or deleted. The visibility of the classes can also be changed.

For example, for a drawing of a house with four separate plan layers (first floor, second floor, basement, and roof), assign all bathroom fixtures to a class called “Fixtures.” Make the Fixtures class invisible to print the floor plans without fixtures, and then make them visible to print the bathroom fixtures for each story of the house.



Fixtures class invisible

Fixtures class visible

Class information can be linked to worksheets. Using the house example in the previous paragraph, not only can the plumbing fixtures plan for the house be printed, but a running inventory of the cost for all plumbing fixtures can be kept (see “Using Worksheets” on page 1319).

If Vectorworks Design Series is installed, you can set up the drawing file to have a set of standard classes, which can be automatically assigned to specific types of objects as they are created. See “Design Series Layers, Classes, and Viewport Standards” on page 184.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

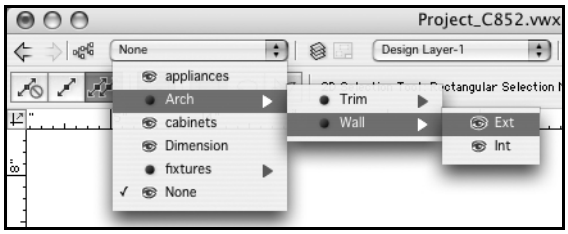
Creating Classes

Decide on a naming scheme before you create classes. If there are a large number of classes, organize them by naming each class with a compound name consisting of up to four parts, separated by a dash. Each name part represents a different level in the class naming structure. For example, a drawing of a building might have a class structure that
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

includes main groups for architecture, plumbing, and electrical objects. Within the architecture group, there might be door, floor, and wall groups. Those groups in turn have subgroups—for example, the wall subgroup might have interior and exterior designations. A class is named according to its position in the class structure, as in Arch-Wall-Ext, Elec-Lite-Ceiling, or Plum-Equip-New.

Class names impact hierarchical display on pop-up menus (including in the Object Info palette, the View bar, and dialog boxes) and in the Organization dialog box and (for Vectorworks Design Series products) the Navigation palette. To enable or disable hierarchical display on pop-up menus, see "Session Preferences" on page 52; for the Organization dialog box and Navigation palette, see "Displaying Classes in Hierarchical Order" on page 159.

In the classes list on the View bar with hierarchical display enabled, each main group is a menu option, with submenus for the subgroups. In the following example, the Arch option has a Wall submenu, with Ext and Int options. This type of organization makes it easy to assign classes as objects are created.



Classes can be created as new, or imported from other current version Vectorworks files or standard files.

To create a new class:

1. For convenience, a new class can be added from multiple locations in the Vectorworks software.
   - **New** button on the Classes tab of the Organization dialog box (**Tools > Organization**)
   - **Classes** button on the View bar to open the Organization dialog box
   - **New Class** option from the Classes drop-down list on the View bar
   - **Class** field on the Shape tab of the Object Info palette
   - **New** context menu on the Classes tab of the Navigation palette (Vectorworks Design Series required)
2. From the New Class dialog box, create a new class, or import a class and its properties from standard or existing Vectorworks files.

Class Type	Action
Create New Class	Creates a class based on current Attributes palette settings; enter a class **Name**
Import Classes	Imports classes and their attributes from standard files or existing files. Files located in the Standards folder, as well as existing files selected previously, are displayed in the list. Select a file; the available classes and descriptions are listed beneath the file name. Select the desired class(es). To select multiple classes from the import list, hold the Ctrl (Windows) or Command (Mac) key while you click.
Show only classes that are not in the current document	If a class name in the current file matches a class in the import file, normally that class is not included on the list as an import option. To display all the classes to be imported, deselect this option. The existing classes in the file are replaced by any imported classes with the same name, changing the existing class definitions (and any associated objects) to those of the imported classes.
Choose	Click **Choose** to select a file for class import. Files must be in the current version.
Creation Options	

Class Type	Action
Saved View Visibility	Sets the visibility of the new class in saved views (when saved views exist in the drawing)
Viewport Visibility	Sets the visibility of the new class in viewports (when viewports exist in the drawing)
Edit Properties After Creation	Immediately after creation, opens the Edit Class(es) dialog box to set the properties of the new class(es)

3. Click **OK** to create the new class(es).

   The classes display in the Classes list on the Organization dialog box, View bar, Object Info palette, and (for Vectorworks Design Series) the Navigation palette. When a new class is created, it does not automatically become the active class.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Classes
Assigning Objects to Classes and Layers
Importing Drawing Structure from Standards or Other Files
Displaying Classes in Hierarchical Order
Setting Class Properties

Setting Class Properties

Once created, the classes display on the Classes tab of the Organization dialog box, where various properties can be set and edited.

To edit classes:

1. Select **Tools > Organization**. Alternatively, click the **Classes** button on the View bar.

   The Organization dialog box opens.

2. Select the Classes tab.

   A list of the current classes in the drawing displays. Depending on which view option is selected at the top of the dialog box, either details or visibilities of the classes display. The visibility of classes in the drawing area and the active class can be changed in **Details** view. The visibility of classes in viewports and in saved views can be changed in **Visibilities** view (see “Setting Visibilities in the Organization Dialog Box” on page 193).

3. To change other class properties, select one or more classes and click **Edit** to open the Edit Class(es) dialog box.

Click to show/hide the parameters.

Parameter	Description
Name	Displays the name of the class; edit as needed
Description	Click to enter additional information about the class. The description displays as a screen tip when you position the cursor over the class name in the Organization dialog box, Navigation palette, or (on Mac only) the Classes list on the View bar. Descriptions also display in the New Class dialog box, so that you can see descriptions of classes before you import them.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Use at Creation	Applies the graphic attributes displayed in this dialog box when you create an object assigned to this class. If **Use at Creation** is not selected, you can apply these attributes to the object later by selecting **Class Style** from the Attributes palette. See "Setting Class Attributes" on page 181 for more information about graphic attributes.
Fill	
Style	Select one of the following fill styles from the list
None	No fill is applied to the objects in this class If a Renderworks texture will be applied to objects in this class, select a fill style. If the object fill is None, no texture displays when the objects are rendered.
Solid	Applies a solid fill to the objects in this class; click the **Color** box to select the desired color
Pattern	Applies a patterned fill and color to objects in this class. Click the **Pattern** box to select the desired pattern, and then select the foreground color and background color from the color boxes below the pattern.
Hatch	Applies a hatch fill to objects in this class. Select a hatch from either the default content or the current file's content.
Tile	Applies a tile fill to objects in this class. Select a tile from either the default content or the current file's content.
Gradient	Applies a gradient fill to objects in this class. Select a gradient from either the default content or the current file's content.
Image	Applies an image fill to objects in this class. Select an image from either the default content or the current file's content.
Opacity	Specifies the class transparency (on Windows, the GDI+ imaging Vectorworks preference must be enabled). Drag the slider to the left to increase the transparency, or enter an opacity percentage in the box to the right of the slider.
Pen	
Style	Select one of the following pen styles from the list
None	No pen outline is applied to the objects in this class
Solid	Applies a solid outline to the objects in this class; click the **Color** box to select the desired color
Pattern	Applies a patterned outline and color to objects in this class. Click the **Pattern** box to select the desired pattern, and then select the foreground color and background color from the color boxes below the pattern.
Line Type	Applies a line type outline to objects in this class. Select a line type from either the default content or the current file's content.
Thickness	Select the line thickness for the class; to use a custom thickness, select **Set Thickness** from the line thickness list to access the Set Thickness dialog box.
Markers	Select the marker style for each end of lines, dimensions, arcs, polylines, 2D polygons, or freehand lines in this class. To use a custom marker style, select **Custom** and specify the marker settings, or select **Edit Marker List** from the marker style list to set the available marker types (see "Editing the Marker List" on page 1103).

Parameter	Description
Use Text Style __ at Creation	From the list, select a text style to apply to objects in this class. To apply the text style when you create an object in this class, also select the **Use Text Style at Creation** option. If **Use Text Style at Creation** is not selected, you can apply this text style to the object later by selecting <Class Text Style> as the **Text Style** in the Object Info palette.
Walls, Roofs, and Other tabs	If Renderworks is installed, click these tabs to set the texture/surface hatch properties for wall, roof, and other objects assigned to the class. See “Applying Textures to Symbols, Walls, and Roofs” on page 1542 for more information.
Saved Views	If there are saved views in the drawing, opens the Saved View Visibilities dialog box. Set the visibility for the new class(es) in the saved views (Visible, Invisible, Gray, or Don’t Save). See “Setting Visibilities in the Organization Dialog Box” on page 193.
Viewports	If there are viewports in the drawing, opens the Viewport Visibilities dialog box. Set the visibility for the new class(es) in the viewports (Visible, Invisible, Gray, or Don’t Save). See “Setting Visibilities in the Organization Dialog Box” on page 193.

4. Click **OK** to return to the Organization dialog box. If objects in an edited class already exist in the drawing, and the class is set to use attributes “at creation,” you are prompted to specify how to apply the changes to the existing objects.
5. Click **OK** from the Organization dialog box to save the changes.

If multiple classes are simultaneously selected for editing, and some or all of the attribute values are different for the selected classes, the editing fields for those values indicate that the value is unknown. When the **OK** button is clicked, the currently defined settings shown are applied to all of the selected classes. Any information with an unknown setting is not applied.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Displaying Classes in Hierarchical Order
Setting Class Attributes
The Attributes Palette
Resource Libraries

Setting Class Attributes

There are two categories of attributes available for each object: object attributes and class attributes. Object attributes are assigned directly to an object from the Attributes palette, Object Info palette, or Resource Browser (depending on the type of attribute). Class attributes are determined by the object’s class settings.

The attributes that an object uses when it is created are controlled by options in the Edit Class(es) dialog box:

- **Use at Creation** (for 2D graphic attributes)
- **Use Text Style at Creation** (for text in text objects, dimensions, callouts, and other annotation objects)
- **Use Textures/Surface Hatches at Creation** (for textures in walls, roofs, and other 3D shapes)

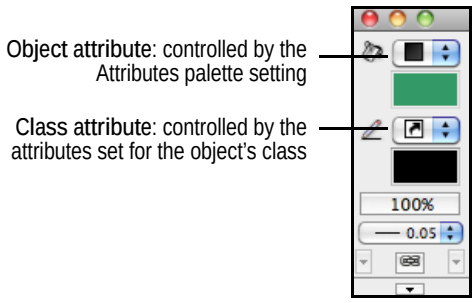
When one of these “use at creation” options is selected for a class, objects created in that class or subsequently assigned to that class use the class attributes. The Attributes palette displays an arrow to indicate that the attributes have been set by class. If objects in that class already existed before the “use at creation” option is selected, you are prompted to decide how to apply the attributes to the existing objects.

Class attributes can also be assigned after objects are created:

- From the Attributes palette, select **Class Style** or **Class Thickness** from the appropriate attribute list.
- In the Shape tab of the Object Info palette, select <Class Text Style> from the **Text Style** list.
- In the Render tab of the Object Info palette, select Class Texture from the **Texture** list.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To override class attributes, select the object(s) and apply different attributes directly from the Attributes palette, Object Info palette, or Resource Browser.

Several plug-in objects (such as Vectorworks Design Series doors and windows) offer the option to control the appearance and visibility of the smaller object parts either individually, or by the same class as the overall object. For example, select the <Door Class> option for the door jamb, lintel, and threshold to assign those door parts to the same class as the door object. If you change the door's class later, the appearance and visibility of the smaller door parts automatically change according to the new class.



Class attributes are represented by a curved arrow

For more information, see "The Attributes Palette" on page 1093.

Setting the Active Class

To be able to remove or edit objects in a particular class, either the class must be active or the class options must be set to allow modifications to other classes (see "Setting Class and Design Layer Options" on page 193). There are several ways to change the active class.

If there are a small number of classes, switch between classes with the **Switch active layer/class** shortcut key combination specified in the Vectorworks preferences (see "Edit Preferences" on page 49). This selects a class by moving up or down through the class list one layer at a time. If the drawing has a large number of classes, use one of the following options.

Setting the Active Class in the Organization Dialog Box

To set the active class:

1. From the Organization dialog box, select the Classes tab in **Details** view.

   The active class is indicated by a check mark to the left of the **Class Name**. The name of the class also is highlighted in bold text.

2. To make a different class active, click the column to the left of its name.
3. Click **OK**.

   The dialog box closes and the active class displays.

Setting the Active Class in the View Bar

To set the active class:

1. Click the Classes list on the View bar to display a list of all of the drawing's classes.

   On Mac, the active class is indicated by a check mark; on Windows, the class name is highlighted in bold text.

2. Click the class to be activated.

   The classes list closes and the active class displays.

Setting the Active Class in the Navigation Palette

To set the active class:

1. From the Navigation palette, select the Classes tab.

   The active class is indicated by a check mark to the left of the class name. The name of the class also is highlighted in bold text.

2. To make a different class active, click the column to the left of its name.

   Alternatively, Right-click (Windows) or Ctrl-click (Mac) on the class to be activated and select **Activate** from the context menu.

Setting the Active Class in the Document Context Menu

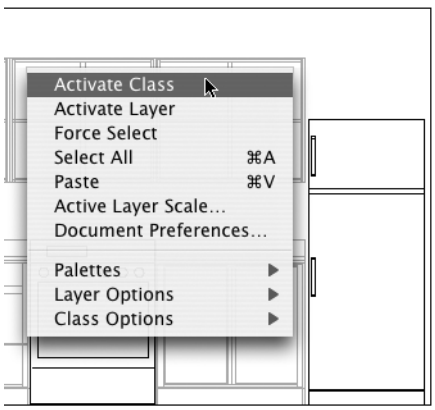
If multiple classes are set to be visible, and the class options are set to show those other classes, the drawing area may display objects that are in non-active classes. Use the **Activate Class** command to make the class of one of these objects active.

The **Force Select** command on the document context menu also changes the active class and the active layer (if necessary), and selects the clicked object.

To set the active class:

1. In the drawing area, right-click (Windows) or Ctrl-click (Mac) a visible object in a non-active class.
2. From the document context menu, select **Activate Class**.

   The object's class becomes active.



Right-click (Windows) or Ctrl-click (Mac) on a cabinet in a gray, non-active class to open the document context menu

The Activate Class command activates the cabinet's class, and the objects that are not in that class become grayed

Copying and Pasting Classed Objects

Objects in a class can be copied and pasted from one drawing to another, even if the destination drawing does not contain the same class as the original drawing. The program automatically creates a new class in the destination drawing and transfers all of the class information from the source drawing. If the destination drawing already has a class with the same name as the source drawing, only the object information is pasted. All of the class information for the destination drawing's class remains unchanged.

When you paste objects that could become invisible due to class visibility settings, specify whether the invisible classes should be made visible so that the pasted objects can be seen.

A L Design Series Layers, Classes, and Viewport Standards

The standards for the Vectorworks Architect and Landmark products take advantage of layer and class characteristics.

Class Characteristics

Each drawing object is assigned to a class, as well as a layer. The class determines the object's appearance, while the layer determines the object's location. Classes apply to the entire file and control the visibility of objects. The currently active class is visible; but classes can be set to be visible, invisible, or grayed when they are inactive. Complex objects, such as symbols or plug-in objects, may contain more than one class; if so, different parts of the object can be hidden or shown. Classes can also be used to assign graphical attributes, textures, and text styles to objects.

Many plug-in objects that are included with Vectorworks Architect and Landmark products are set with pre-assigned classes. The appropriate classes are created at setup and by certain other commands (see "Automatically Created Classes" on page 185). The use of automatic classing is determined with the **Standard Naming** setup command. For more information, see "Importing Drawing Structure from Standards or Other Files" on page 161 and "Classes" on page 176.

Layer and Viewport Characteristics

A layer is a named container that holds items. There are two types of layers: design layers and sheet layers. Design layers are used for drawing and modeling the elements of a project. Sheet layers are created for the presentation of a finalized project, and can contain viewports, title blocks, notes, and other annotations. A viewport, located on a sheet or design layer, is a particular combination of visible, grayed, and/or hidden design layers and classes.

Layers have certain characteristics that are used when drawing and structuring files:

- Design layers can automatically set default elevation values for objects they contain. They create natural structural divisions within a project for objects on different floors or different vertical locations within a floor.
- Design layers can be visible, invisible, or grayed. Sheet layers are always set to Active Only.
- Design layers, as well as viewports, can be displayed at different drawing scales, for the display of all aspects of a project plan from the site model to details.
- Design layers, as well as viewports, can have different 3D views. A building can be viewed in Plan view in one viewport and in an elevation or perspective view in another.
- Layers can be contained in different files and shared using workgroup referencing.

Projects set up according to standards contain both design layers and sheet layers with viewports. An architectural project file contains, at a minimum, stories with design layers for every level, as well as a number of viewports on sheet layers.

A typical Vectorworks Landmark product standard file setup includes landscape site plans composed of shared model information on four layers:

- Mod-Site-Arch – contains any buildings or other improvements
- Mod-Site-Civil – contains topographic and survey information
- Mod-Site-DTMData – contains the site model output
- Mod-Site-Landscape – contains tree and planting data

When a file is set up with the **Create Standard Viewports** command (Vectorworks Architect required), the appropriate classes and layers are created automatically. The number and types of layers and classes created depend on the setup selections. In the Vectorworks Architect product, design layers are created by stories, and begin with "Mod-" (model layers, since this is where the model is designed). The **Create Standard Viewports** command creates the appropriate viewports and sheet layers for the viewports (beginning with "Sheet-"), along with the appropriate classes if they are not already in the file (see "Standard Viewports" on page 186).

The **Standard Naming** command establishes or changes the naming conventions used for these classes, design layers, sheet layers, and viewports or saved views (see "Standard Naming" on page 92).

Set up a new, blank file with standards, and then save as a template for future use.

Setup standards are determined by the LayerMap.G worksheet, which can be imported and customized by advanced users. If an existing file already contains a set of custom standards and the LayerMap.G worksheet is present in the file, the Import LayerMap.G dialog box opens when selecting **Create Standard Viewports** (Vectorworks Architect required). Select whether to keep the imported and customized worksheet, or whether to revert to the standard setup. See "Using the Layermap Worksheet" on page 1891 for more information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Automatically Created Classes
Standard Viewports

Automatically Created Classes

Several classes are created automatically by features in the Vectorworks Design Series products, regardless of whether standards have been established at setup. See "Setting Class Properties" on page 179. Examples of automatically created classes include:

- NonPlot (Vectorworks Architect): This class is created as part of doors and windows. The loci that define the window and door edges are created in this class, which is normally set to Invisible so that the loci are hidden.
- Redlines (Vectorworks Architect, Landmark, Spotlight): This class is created by the **Redline** tool. All redline objects are placed in this class, which allows all redlines in the file to be shown or hidden. This class is toggled to visible and invisible by the **Show or Hide Redlines** command.
- Guides (Vectorworks Architect and Landmark): This class is created and used by selecting **Modify > Guides > Make Guide**.
- The **Wall Framer** command (Vectorworks Architect) creates the following classes: Framer-Block, Framer-Sole Plate, Framer-Header, Framer-Stud, Framer-Sill, and Framer-Top Plate.
- Site-DTM-Modifier (Vectorworks Architect and Landmark): This class is created by the Pad, Texture Bed, Grade Limits, and Spoil Pile objects. The Landscape Walls and Roadway objects include pad and grade limit objects if the **Use Site Modifiers** and **Use Grade Limits** check box is selected on the Object Info palette. This class is toggled to visible and invisible by the **Show or Hide Site Modifiers** command.
- Irrigation-SprayPat (Vectorworks Landmark): This class is created by using the Irrigation Head and Drip Emitter objects. This class is toggled to visible and invisible by the **Show or Hide Spray Pattern** command.

Object Auto-classing

Auto-classing is the automatic assignment of certain objects to a default class. Many plug-in objects in the libraries provided have been pre-assigned to the proper class according to the drawing standard for the Vectorworks Architect and Landmark products (VWArch). For a list of auto-classing objects, see "Auto-classing Objects" on page 1895.

If the **Use Auto-classing** check box is selected in Standard Naming (see "Standard Naming" on page 92), then these plug-in objects will be automatically placed in the designated class as they are added to the drawing. The object's class is created automatically if it does not yet exist.

If a file has not been structured according to setup standards, or the **Use Auto-classing** check box is not selected, the objects are placed in the active class. The objects, upon regeneration, are assigned to the proper class if the file is later set up. Any symbol, when created, can be set to default to a class from the Symbol Insertion Options dialog box.

If you use a naming standard other than VWArch, the default class of the object libraries must be reset.

Save a backup version of the object libraries before editing them. See "Resource Libraries" on page 219.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To set the default class of all the symbols in a library file:

1. Select **File > Open**.

   The standard Open dialog box opens.
2. Select the Libraries folder, and then click **Open**.
3. Select the first object library file to convert, and then click **Open**.

   The selected file opens in the drawing window.
4. Select **Tools > Utilities > Set Default Symbol Class**.

   A warning dialog box opens. Click **Yes** to acknowledge converting all symbol definitions in the file to the new default class name.

   The Enter String dialog box opens.
5. Enter the default class name for the symbols, and then click **OK**.

   Ensure that the name is spelled correctly to match the desired custom class standard. This command can be undone if necessary. If the name matches an existing name in the file other than a class, an alert dialog box opens.
6. Select **File > Save** to save the changes.
7. Repeat steps 1 through 6 for each object library.

   Any time that symbols are used from this file, they will take on the specified class as their default class.

   The command does not distinguish between one symbol definition and another. All symbol definitions in the file will take on the new class name. For that reason, to use on custom libraries, run the command on a copy of the file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Auto-classing Objects

Standard Viewports

The **Create Standard Viewports** command creates standards-compliant viewports and their associated sheet layers, with the layer and class visibilities of a standard drawing. See “Creating Sheet Layer Viewports” on page 1616.

If desired, views corresponding to the viewports can also be created. This allows easy navigation through the different drawing views (with set layer and class visibilities) during the design process.

Viewports are created in five categories. The available viewport types are:

- Site Plan Drawings
- Project Plan Drawings
- Floor Plan Drawings
- Auxiliary Plan Drawings
- Notation Drawings

  Select the **Create Standard Viewports** command again to make changes to the project settings at any time. The column on the right shows the current viewports when the command is run again.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Standard Viewport Preferences
Creating Standard Viewports

Setting Standard Viewport Preferences

Stories must be created before creating standard viewports (see “Setting Up the Building Structure with Stories” on page 172). The default viewport scale and sheet border settings can be set prior to adding standard viewports to a project.

To set viewport preferences:

1. Select **File > Document Settings > Create Standard Viewports**.

   The Create Standard Viewports dialog box opens.

2. Click **Preferences**.

   The Create Viewport Preferences dialog box opens. The preferences apply as viewports are added to the list for inclusion in the project.

3. Set the default scale for each type of viewport; select a sheet border and, if desired, title block to add automatically to each sheet layer. The scale settings affect only the viewport scale, not the layer scale of any model layers. See “Adding a Sheet Border” on page 77 for more information on sheet borders.

4. Click **OK** to return to the Create Standard Viewports dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Standard Viewports
Standard Viewports

Creating Standard Viewports

Stories must be created before creating standard viewports (see “Setting Up the Building Structure with Stories” on page 172).

To create standard viewports:

1. Select **File > Document Settings > Create Standard Viewports**.

   The Create Standard Viewports dialog box opens. Select a drawing category from the **Type of Drawing** list. The available drawing types display in the **Drawing Types** list on the left, with a short description beneath. Select the viewport to be created in the **Drawing Types** list and then click **Add** to move it to the **Viewports to Be Created** list on the right.

   For auxiliary view viewports (sections and elevations), types with a -Man suffix (such as Sections-Man) typically indicate that the elements of the section or elevation are to be drawn manually on, for instance, Mod-Section or Mod-Elevation layers, which are created along with the viewports. The visibility of all other layers is set to Invisible for these viewports. Types with a -VP suffix (such as Sections-VP) are for creating a view or a section viewport of the model from existing Mod- layers. No new layers are created and the visibility of all existing Mod- layers is set to Visible for these viewports.

Click to show/hide the parameters.

Parameter	Description
Type of Drawing	Lists the categories of viewport types
Drawing Types	Lists the available viewport types in the selected **Type of Drawing** category
Description of Drawing Type	Provides a description of the **Type of Drawing** category
Viewports to be Created	Lists the viewports that will be created, along with the specified scale, sheet border and sheet layer parameters
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Add >	Adds the selected viewport types from the **Drawing Types** list to the **Viewports to be Created** list, including them in the drawing setup
< Remove	Removes the selected viewport from the **Viewports to be Created** list
<< Remove All	Removes all viewports from the **Viewports to be Created** list
Preferences	Specifies the scale and sheet border parameter preferences when adding viewports to the **Viewports to be Created** list
Sheet Options	
Create All Stories (Vectorworks Architect required)	For Floor Plan drawings, creates the selected viewport for each story; deselect to specify an individual story based on its **Suffix** instead
Suffix	For Floor Plan, Auxiliary or Notation drawings, specify the individual story by its suffix to create the viewport
Use sheet border on selected sheet	Places the sheet border specified in Create Viewport Preferences on the sheet layer when the viewport is added to the drawing
Use Vectorworks uniform drawing numbering system	Uses the Vectorworks uniform sheet numbers and sheet type designators
Use automatic drawing coordination	Matches the sheet number and title to the sheet border title block
Viewport Options	Select a viewport from the **Drawing Types** list to set its parameters before adding it to the **Viewports to be Created** list. Alternatively, select a viewport in the **Viewports to be Created** list to set or change its options.
On Sheet Layer	Select a sheet layer where the viewport will be placed. A default sheet layer name is provided, but a different sheet layer can be selected. Alternatively, a new sheet layer name can be created, by selecting New Sheet Layer and providing the sheet layer name.
Viewport parameters	Sets the viewport scale, view, projection, perspective, and rendering as described in “Creating a Sheet Layer Viewport from a Design Layer” on page 1616
Create corresponding view for each viewport	For each viewport in the **Viewports to be Created** list, creates a corresponding view with the same name. It is useful during the design process to navigate among the different project views, where the proper layer and class visibilities have already been set.

2. Once the list of viewports is ready, click **OK**. Any sheet layers specified that do not already exist in the drawing are created along with the listed viewports (and views if the **Create Corresponding View** option was selected).

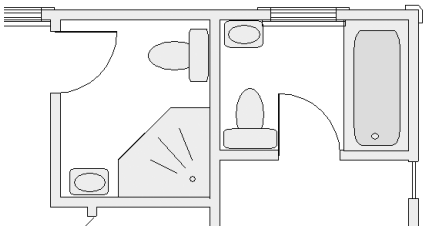
   In a new drawing, the viewports display with a red X, indicating that they are currently empty. As the drawing is developed on the design layers, the viewports will display the contents appropriately. Depending on the rendering mode specified, some viewports may require updating with the **Update Selected Viewports** command.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

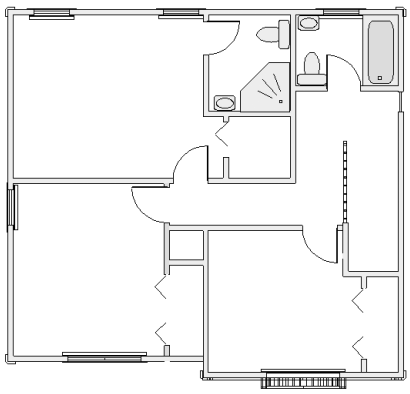
Standard Viewports
Standard Naming
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Saved Views

A saved view is like a camera that is set up to show a drawing from a certain orientation, with a specific set of viewing parameters, including which class and design layer are active, the visibilities of the inactive classes and the design layers, the current zoom and pan, and the page location. If Vectorworks Design Series is installed, the plan rotation and the clip cube position can be saved.



Zoomed in view



Zoomed out view

Views are also used to create Move Along Path animations (see “Creating and Editing Move Along Path Animations” on page 1182 for more information).

Saved views can be created, edited, duplicated, and deleted from the Organization dialog box as described in the following sections.

Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Saved Views

To save the current drawing area view:

1. Select **View > Save View**.

   Alternatively, from the Organization dialog box, click the Saved Views tab, and then click the **New** button. If Vectorworks Design Series is installed, from the Navigation palette, click the Saved Views tab, and then select the **New** command from the navigation menu.

2. The Save View dialog box opens.
3. Specify the view options, the active layer and class, and the visibilities of layers and classes.

Click to show/hide the parameters.

Parameter	Description
View Name	Specify the view name
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Save View Orientation	Saves the general view parameters of the active layer, including the projection, 3D orientation, and page origin settings. If Vectorworks Design Series is installed, the plan rotation and the clip cube position are also saved.
Save Zoom and Pan	Saves the zoom and pan settings of the active layer
Save Page Location	Saves the design layer page location with the view; if deselected, the current page location setting is used when the view is displayed. The page location of sheet layers cannot be saved.
Save Unified View	Saves the unified view status with the view; if deselected, the current unified view setting is used when the view is displayed
Save Render Mode and Options	Saves the render mode and options with the view; if deselected, the current render mode and options are used when the view is displayed Saving the render settings with the view is an effective method of saving and restoring custom rendering options.
Save Layer Visibility	Specifies the layer visibility options to save; if deselected, the current layer visibility settings are used when the view is displayed
Layer Options	Select the design layer display options
Active Layer	Select the active layer; if a sheet layer is selected, the **Layers** button is disabled
Layers	Opens the Layer Visibilities dialog box; specify the design layer visibilities for the saved view (see “Using Visibility Columns” on page 193 for details)
Save Class Visibility	Specifies the class visibility options to save; if deselected, the current class visibility settings are used when the view is displayed
Class Options	Select the class display options
Active Class	Select the active class from the list of classes
Classes	Opens the Class Visibilities dialog box; specify the class visibilities for the saved view (see “Using Visibility Columns” on page 193 for details)

4. Click **OK** to save the view with the specified settings. The saved view is then available from the **Saved Views** menu and from the Organization dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Saved Views
Creating or Editing Saved Views Using the Saved Views Menu
Viewing History
Setting Class and Design Layer Options
Setting Visibilities in the Organization Dialog Box

Editing Saved Views

Set the active class and layer, the class and design layer options, and the class and design layer visibilities when you create the saved view (in the Save View dialog box). Those initial settings can be changed later from the Organization dialog box.

To edit a saved view:

1. Select **Tools > Organization**. The Organization dialog box opens.
2. Select the Saved Views tab in **Visibilities** view.

   The visibilities of classes and design layers for the selected saved view display.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Select a view to edit from the **Saved View Name** list.
4. If **Save Class Visibility** was selected in the Save View dialog box, **Class Options** and **Active Class** are enabled in the Organization dialog box. If **Save Layer Visibility** was selected in the Save View dialog box, **Layer Options** and **Active Layer** are enabled in the Organization dialog box. Change the **Active Layer** and the **Active Class** as necessary. Change the **Class Options** and **Layer Options** as described in “Setting Class and Design Layer Options” on page 193.
5. Change the visibilities of classes and design layers as necessary. See “Setting Visibilities in the Organization Dialog Box” on page 193.
6. To change other saved view properties, click **Edit**.

   The Edit Saved View dialog box opens.

   The settings are the same as when the view is created. Classes and layers that were added after a view was created are listed as visible in the visibility settings.

   If the layer or class visibility was saved when the view was created, **Restore Layer Visibility** and **Restore Class Visibility** are enabled. Select **Restore Layer Visibility** to restore the layer visibilities, the layer options, and the active layer that were set when the view was saved. Select **Restore Class Visibility** to restore the class visibilities, the class options, and the active class that were set when the view was saved.

   Saved views are saved as VectorScript macros. If necessary, click **Edit Script** to edit the script.
7. Click **OK** to save the changes. Click **OK** again to close the Organization dialog box.

Another way to edit a saved view is through the Saved Views palette. Select **Window > Script Palettes > Saved Views**. Press the Option (Mac) or Alt (Windows) key and double-click the view name to edit. Double-click the view script name to switch the current drawing area to the saved view.

If Vectorworks Design Series is installed, you can also edit saved views from the Navigation palette; see “The Navigation Palette” on page 199.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Saved Views
Creating or Editing Saved Views Using the Saved Views Menu

Redefining Saved Views

To change the content of a saved view, use the **Redefine** command. This changes the saved view settings to match those of the current drawing area view, including the current layer options and class options, the plan rotation, the visibility of layers and classes that are inactive, and the active layer and class.

To redefine a saved view:

1. From the Navigation palette, select the Saved Views tab.
2. Select the view to be changed from the list.
3. From the Navigation menu or the view context menu, select **Redefine**.

   The Redefine Saved View dialog box opens. Specify the view options and the visibility parameters.

Click to show/hide the parameters.

Parameter	Description
View Name	Specify the view name
Restore Options	
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
View Orientation	Saves the general view parameters (current projection, 3D orientation, plan rotation, and page origin settings) of the active layer
Zoom and Pan	Saves the zoom and pan settings of the active layer
Page Location	Saves the design layer page location with the view; if deselected, the current page location setting is used when the view is displayed. The page location of sheet layers cannot be saved.
Unified View	Saves the current unified view status, whether on or off; if deselected, the current unified view status is ignored and the status in effect when the view was created is used
Render Mode and Options	Saves the current render mode and render options; if deselected, uses the render mode and options that were set when the view was created (if saved at that time)
Layer Visibility	Saves the current layer visibilities; if deselected, uses the visibilities that were set when the view was created (if saved at that time)
Class Visibility	Saves the current class visibilities; if deselected, uses the visibilities that were set when the view was created (if saved at that time)

4. Click **OK** to save the current drawing area view with the specified settings.

Creating or Editing Saved Views Using the Saved Views Menu

The View bar has shortcuts to save a view, to edit a saved view, or to switch the current drawing area view to a previously saved view.

> Views can also be accessed through the Saved Views palette. Select **Window > Script Palettes > Saved Views**. Double-click the view name to switch to that view.

To use the **Saved Views** menu:

1. Click the **Saved Views** menu on the View bar.

2. Select the desired item from the menu.

Menu Item	Description
Save View	Opens the Save View dialog box
Edit View	Opens the Saved Views tab of the Organization dialog box
List of saved views	Select a saved view from the list to switch to that view

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Saved Views
Editing Saved Views
Viewing History
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Class and Design Layer Options

The **Class Options** and **Layer Options** commands control how all the classes or design layers in a drawing display. For example, a drawing project can be set to display only the active class, temporarily hiding all objects assigned to other classes.

To change the options for displaying, snapping to, and editing objects in the current class or design layer:

1. Select **View > Class Options** (or **Layer Options**) and then the option.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) in the drawing area and select **Class Options** or **Layer Options** from the document context menu.

Command	Description
Active Only	Displays only objects in the active class/layer; only the active class/layer prints
Gray Others	Displays the active class/layer normally and all other classes/layers appear dimmed (except for those set to invisible); even though visible, objects in dimmed classes/layers cannot be edited
Gray/Snap Others	Displays the active class/layer normally and all other classes/layers appear dimmed (except for those set to invisible); objects in any normally displayed or gray class/layer can be snapped to. Only objects in the active class/layer can be edited.
Show Others	All classes/layers display normally, except for those set to invisible or grayed; even though visible, objects in classes/layers other than the active class/layer are not editable and cannot be snapped to
Show/Snap Others	All classes/layers display normally, except for those set to invisible or grayed; objects in any normally displayed or gray class/layer can be snapped to. Only objects in the active class/layer can be edited.
Show/Snap/ Modify Others	All classes/layers display normally, except for those set to invisible or grayed. Objects in any normally displayed or gray class/layer can be snapped to; only objects in normally displayed classes/layers can be edited. (An object on another layer can only be edited if its layer scale and view are the same as those of the active layer.) Locked objects display with gray highlighting.

2. The current class or design layer display changes accordingly.

   Click here for a video tip on this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Visibilities in the Organization Dialog Box
The Visibility Tool

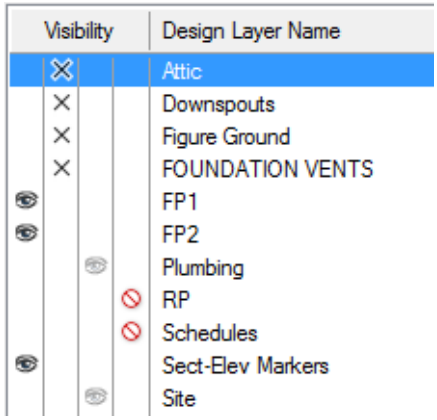
Setting Visibilities in the Organization Dialog Box

The active class and design layer are always visible. Each inactive class and design layer can be set to be visible, invisible, or gray. These visibilities can be set independently for the drawing area, saved views, and viewports.

The level of gray for grayed layers and classes can be adjusted for printing; see “Printing a File” on page 1763.

Using Visibility Columns

For maximum usability, class and design layer visibilities can be set in multiple places in the Organization dialog box, and also in other dialog boxes. The **Visibility** columns work the same wherever they are used.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



To change the setting for a single class or design layer, click in one of its visibility columns. To change the settings for multiple items, click a visibility column as follows:

- Press the Ctrl key (Windows) or Cmd key (Mac) and click selected class or layer rows
- Press the Shift key and click the first and last rows of a group of classes or layers
- Press the Alt key (Windows) or Option key (Mac) and click any row to change all classes or layers

For classes displayed in hierarchical order, change the visibility setting of a group header to change the visibility of all of its subclasses; see “Displaying Classes in Hierarchical Order” on page 159.

Column	Description
Visible	Class/design layer is visible; objects in this class/layer display when another class/layer is active
Invisible	Class/design layer is invisible; objects in this class/layer display only when the class/layer is active
Gray	Class/design layer is gray; objects in this class/layer are dimmed when another class/layer is active
Don’t Save	For saved views, a fourth column displays to the right of the other columns. When selected, class/design layer visibility is not saved for the saved view; the current class/layer visibility is used when the view is displayed.

Setting Class and Design Layer Visibility for the Drawing Area

Use the Organization dialog box to set the visibilities of classes and design layers in the drawing area. If Vectorworks Design Series software is installed, you can use the Navigation palette to set visibilities also (see “The Navigation Palette” on page 199).

Class and design layer visibility in the drawing area are also affected by the **Class Options** and **Layer Options** settings. See “Setting Class and Design Layer Options” on page 193 for details.

To set the class and design layer visibilities for the drawing area:

1. From the Organization dialog box, select the Classes or Design Layers tab in **Details** view.
2. Change the **Visibility** settings as desired.
3. To see the changes before saving them, click **Preview**.
4. Click **OK** to save the changes.

Setting Class and Design Layer Visibility for Viewports and Saved Views

Use the Organization dialog box to set the visibilities of classes and design layers in existing viewports and saved views. You can also set these visibilities while creating and editing classes, design layers, viewports, and saved views, as described in the following topics:

- "Creating Classes" on page 177 and "Setting Class Properties" on page 179
- "Creating Layers" on page 162 and "Setting Design Layer Properties" on page 165
- "Viewport Properties" on page 1636
- "Creating Saved Views" on page 189

To set the class and design layer visibilities for viewports and saved views:

1. From the Organization dialog box, select a tab in **Visibilities** view.

Organization Dialog Box Tab	Sets Visibility of
Classes	Classes in viewports and saved views; if classes are displayed in hierarchical order and a class group header is selected, the viewport and saved view lists are disabled
Design Layer	Design layers in viewports and saved views
Viewports	Classes and design layers in a viewport
Saved Views	Classes and design layers in a saved view

2. Select one or more class, design layer, viewport, or saved view names and change the **Visibility** settings for classes and design layers as desired.
3. On the Saved Views tab, change the active design layer and class, and the design layer and class options.
4. To see the changes before saving them, click **Preview** (not available on the Saved Views tab).
5. Click **OK** to save the changes.

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Class and Design Layer Options
The Visibility Tool

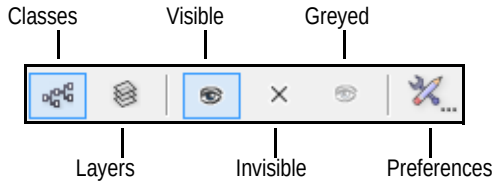
The Visibility Tool

The **Visibility** tool changes the visibility of either the layer or class associated with a clicked object. If you click an object in the drawing area, the document visibility changes. If you click an object in a sheet layer or design layer viewport, only the visibility in that viewport changes (with some exceptions). You can also make rapid changes to the visibilities of multiple layers or classes in the drawing at one time by double-clicking the tool itself.

You cannot change the class and layer visibilities within a section viewport with the **Visibility** tool.

The **Visibility** tool is available in the workspaces of all Vectorworks Design Series products; for the Vectorworks Fundamentals product, you can edit the workspace to add the **Visibility** tool.

Several modes are available, to specify whether to change class or layer visibility, and what type of visibility change to make.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mode	Description
Classes	Affects the visibility of the clicked object's class
Layers	Affects the visibility of the clicked object's layer
Visible	Sets the clicked object's class or layer to visible
Invisible	Hides the clicked object's class or layer
Greyed	Sets the clicked object's class or layer to gray
Preferences	Specifies the behavior when the **Visibility** tool is double-clicked

Click here for a video tip about this topic (Internet access required).

Layer or Class Visibility Changes

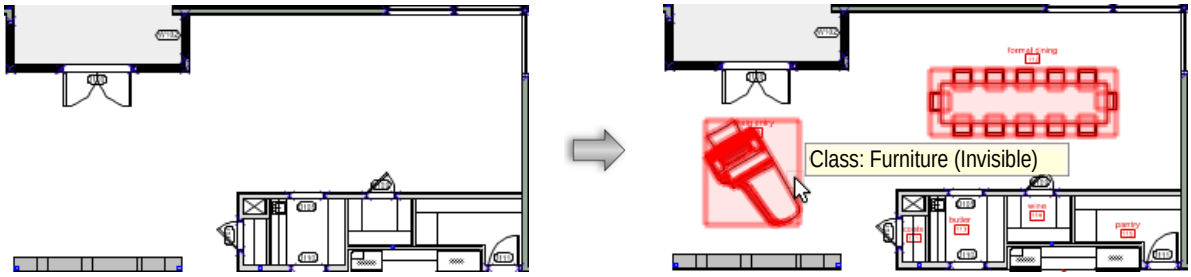
 To change class or layer visibilities based on a clicked object:

1. Click the **Visibility** tool from the Basic palette.
2. From the Tool bar, select **Classes** or **Layers** mode. Then select whether to make the object's class or layer **Visible**, **Invisible**, or **Greyed**.
3. If the class or layer to be changed is currently invisible, ensure that the class or layer options are not set to Active Only. Press and hold the shortcut key for the **Visibility** tool (V by default) to temporarily make all classes (in Classes mode) or layers (in Layers mode) visible. If the cursor is over a viewport, all classes or layers in the viewport become visible.

   In Classes mode, if the cursor is over a design layer viewport that is set to use the class visibilities of the document, all document classes become visible (which includes the classes in the viewport).
4. Move the cursor over a drawing object that is in the class or layer to be changed. All objects that will be affected by the visibility change are highlighted. In addition, a screen tip indicates the name of the class or layer that will be changed, as well as its current visibility setting. If the object is in a viewport, the screen tip also indicates the name of the viewport in which the change will be made.

   In Classes mode, if the object is in a design layer viewport that is set to use the class visibilities of the document, the screen tip does not show the viewport name. This indicates that a click on the object will change the class visibility for the document (not the viewport)—including objects of that class that are not in the viewport.
5. Click on the object to change the visibility setting for its class or layer.

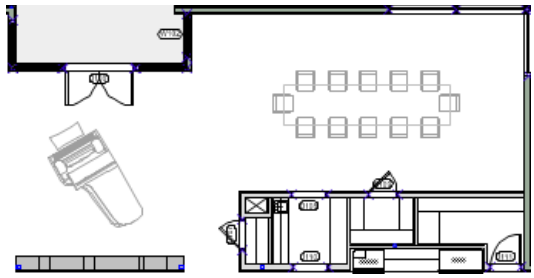
   The object's class or layer is set to visible, invisible, or gray, and the Message bar shows the change that occurred. The Navigation palette (Vectorworks Design Series required) updates to display the new visibility status. On a design layer, changes to class visibility redraw the entire drawing; changes to layer visibility redraw the layer.



With the **Visibility** tool set to make the class gray, press the V key to temporarily show all classes, including those that are currently invisible

Click an object in the “Furniture” class

The “Furniture” class becomes gray



Reasons for Class or Layer Visibility not Changing

When you use the **Visibility** tool to make class or layer changes, several factors may prevent the visibility from changing as expected.

- The layer and class options control the visibility of objects in the inactive layers and classes. For example, if the layer options are set to gray objects in inactive layers, grayed objects do not become fully visible when clicked with the **Visibility** tool. (See “Setting Class and Design Layer Options” on page 193.) An alert displays in this situation.
- The **Visibility** tool cannot hide or gray the active class or layer immediately, since the active class or layer is always set to visible. An alert displays in this situation.
- A container object (such as a symbol or a group) and its component parts can belong to different classes, and their visibilities can be set differently. For example, if the classes of the individual objects that make up a symbol are set to invisible or gray, they do not become visible when the symbol’s class is made visible.
- In a viewport, changes to class or layer visibility display immediately if the viewport has a background render mode of Wireframe, Sketch, or OpenGL. For other render modes, you must update the viewport to see the visibility change. An alert displays in this situation.
- If a design layer viewport (Vectorworks Design Series required) is embedded in another viewport, several factors affect the behavior of the **Visibility** tool. See “Visibility Tool Behavior in Embedded Viewports” on page 198 for more information.

Global Visibility Changes

The **Visibility** tool can make global changes to the class or layer visibilities of a drawing. In a complex drawing, for example, this can be an easy way to quickly show only the classes or layers of selected objects. All classes or layers can easily be set to show again. This feature does not affect visibilities in viewports.

 To make global changes to the visibility of classes or layers:

1. To make changes to the class or layer visibility of selected objects, select the objects first.

   The objects do not need to be in the same class or layer.
2. Click the **Visibility** tool from the Basic palette.
3. Select **Preferences** from the Tool bar.

   The Visibility Tool Preferences dialog box opens. Specify the effect to occur when the **Visibility** tool is double-clicked.

Click to show/hide the parameters.

Parameter	Description
Makes all classes visible	All classes in the drawing area are set to visible
Makes all layers visible	All layers in the drawing area are set to visible
Makes only the classes of the selected objects visible	The class(es) of selected objects remain visible; other classes are set to invisible
Makes only the layers of the selected objects visible	The layer(s) of selected objects remain visible; other layers are set to invisible

4. Click **OK**.
5. Double-click the **Visibility** tool to execute the selected action.

   The classes or layers in the drawing area are set to visible or invisible as specified, and the Message bar shows the change that occurred. The Navigation palette (Vectorworks Design Series required) updates to display the new visibility status.

Reasons for Global Visibility not Changing

When you use the **Visibility** tool to make global changes, several factors may prevent the visibility from changing as expected.

- The layer and class options control the visibility of objects in the inactive layers and classes. For example, if the layer options are set to gray objects in inactive layers, grayed objects do not become fully visible when the **Visibility** tool is double-clicked. (See “Setting Class and Design Layer Options” on page 193.) An alert displays in this situation.
- The **Visibility** tool cannot hide the active class or layer immediately, since the active class or layer is always set to visible. An alert displays in this situation.
- A container object (such as a symbol or a group) and its component parts can belong to different classes, and their visibilities can be set differently. For example, if the classes of the individual objects that make up a symbol are set to invisible or gray, they become visible only when the first double-click option (Makes all classes visible) is used.

Visibility Tool Behavior in Embedded Viewports

Design layer viewports (Vectorworks Design Series required) can be embedded in both sheet layer and design layer viewports, and they can be referenced or not. There are settings in the viewport properties of all of these viewports to control the way classes and layers display, which affect the behavior of the **Visibility** tool.

Cases Where Viewport Visibility Does not Change

The following cases describe the conditions under which the **Visibility** tool cannot change the visibilities of the class or layer of a clicked object, when the object is in a design layer viewport that is embedded in another viewport.

- The top-level viewport is a sheet layer viewport, and it is set to use the class or layer visibilities of the embedded viewport. The tool can only change the top-level viewport's visibility setting for the class or layer of the embedded viewport itself. The tool cannot change the visibility for the class or layer of an object inside the embedded viewport.
- The top-level viewport is a sheet layer viewport, and it is not set to use the class or layer visibilities of the embedded viewport. However, the embedded viewport is referenced. In a referenced viewport, each class can either keep its own settings, or take on the settings of the document in which it is embedded. In Classes mode, if you click on an object whose class is set to keep its own settings, the tool changes the top-level viewport's visibility setting for the class of the embedded viewport itself. In Layers mode, the tool can only change the top-level viewport's visibility setting for the layer of the embedded viewport itself.
- The top-level viewport is a design layer viewport, and the embedded viewport is referenced. In Classes mode, if you click on an object whose class is set to keep its own settings, the tool changes the top-level viewport's visibility setting for the class of the embedded viewport itself.
- The top-level viewport is a design layer viewport. In Layers mode, the tool can only change the top-level viewport's visibility setting for the layer of the embedded viewport itself. The tool cannot change the visibility for the layer of an object inside the embedded viewport.

Cases Where Class Visibility for the Document Changes

The following cases describe the conditions under which the **Visibility** tool changes the class visibilities for the entire document, when you click on an object in a design layer viewport that is embedded in another viewport.

- The top-level viewport is a design layer viewport, and it is set to use the class visibilities of the document. The embedded viewport is not referenced. The tool changes the visibility of the class of the clicked object, not only in the viewport, but in the entire document.
- The top-level viewport is a design layer viewport, and it is set to use the class visibilities of the document. The embedded viewport is referenced. If the clicked object belongs to a class that is set to take on the settings of the document, the tool changes the class visibility for the entire document.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Class and Design Layer Options
Changing the Class Properties of Sheet Layer or Design Layer Viewports

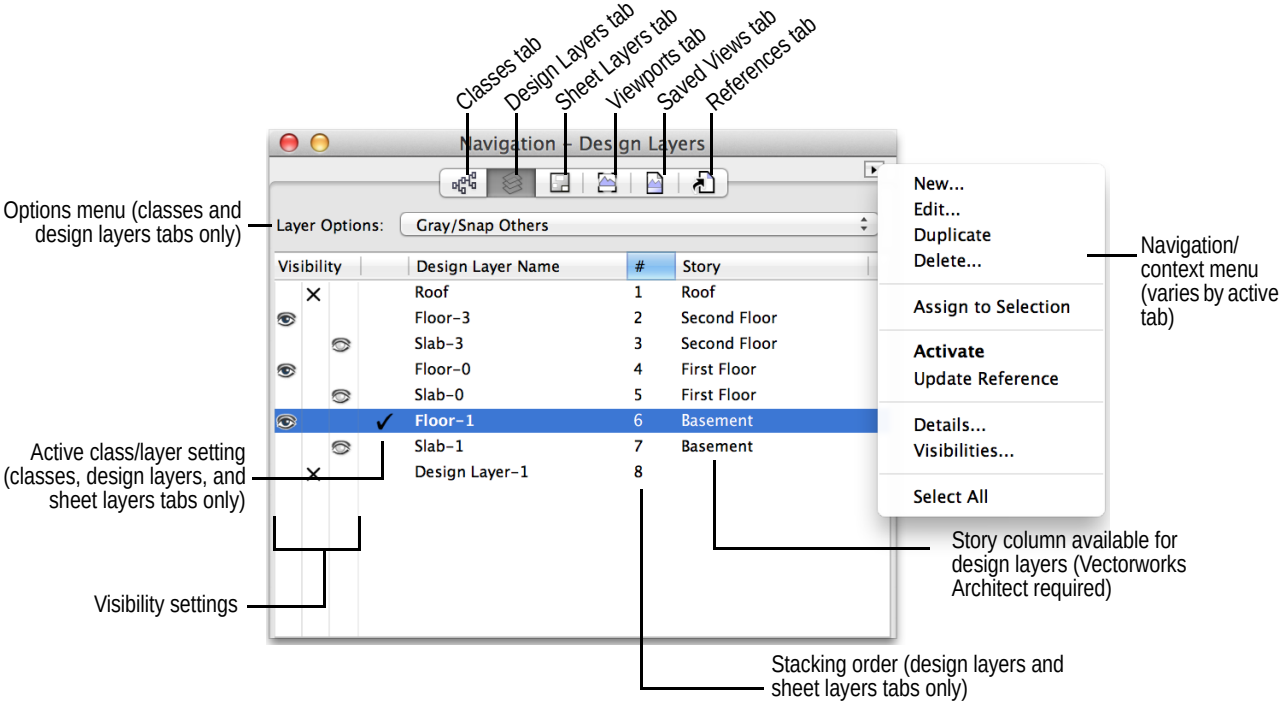
D The Navigation Palette

The Navigation palette provides access to classes, design layers, sheet layers, viewports, and saved views, similar to the Organization dialog box. However, unlike the Organization dialog box, the drawing area is accessible while the Navigation palette is open.

To use the Navigation palette:

1. Select **Window > Palettes > Navigation**.

   The Navigation palette opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click the desired tab to set the active tab upon opening the Navigation palette. Select **Window > Palettes > Save Palette Positions** to save the changes. To revert the active tab back to the setting established when a custom workspace was created (in the User Data and Preferences folder), click **Reset Saved Settings** from the Session tab of Vectorworks preferences (see "Session Preferences" on page 52).

2. Select the appropriate tab: Classes, Design Layers, Sheet Layers, Viewports, Saved Views, or References.

A list of items of that type in the current drawing displays. If a description was entered for a class or layer, the description displays in a screen tip when you position the cursor over the item name; on Windows, click the disclosure arrow on the screen tip to collapse or expand it.

3. Change the settings for classes, design layers, sheet layers, viewports, saved views, or references as needed.
   - Click a class or design layer's **Visibility** column to set the visibility for objects in that layer. To set the same visibility for all classes or design layers in the list, Option-click (Mac) or Alt-click (Windows) one of the Visibility columns.
   - Use the navigation context menu options to toggle the display of classes in hierarchical order, or to expand or collapse the class list for better manageability.
   - If classes are displayed in hierarchical order, click the **Visibility** column for a class group header and all of its sublevels inherit that visibility setting.
   - Double-click a class, design layer, or sheet layer to activate it. Alternatively, click the column to the left of the item's name to activate it.
   - Double-click a saved view to switch to that view.
   - Double-click a viewport to switch the active layer to the sheet layer or design layer that contains the viewport, and to select the viewport (if a viewport is in a class with settings that prevent the viewport from being selected, a message displays to indicate that the active class changed to the viewport class).
   - Select a class or design layer and then select one of the **Class Options** or **Layer Options**.

- Change the stacking order of a design layer or sheet layer by clicking the number in its # column and dragging it up or down the list.
- Double-click on a tab in the Navigation palette to open the equivalent tab in the Organization dialog box.

See the following sections for details:

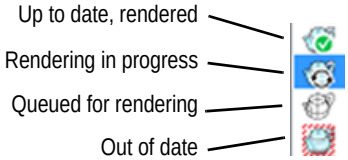
- "List Box Functionality" on page 44
- "Displaying Classes in Hierarchical Order" on page 159
- "Changing the Design Layer Stacking Order" on page 167
- "Setting the Active Layer" on page 164
- "Changing the Sheet Layer Stacking Order" on page 171
- "Creating Classes" on page 177
- "Setting the Active Class" on page 182
- "Saved Views" on page 189
- "Setting Visibilities in the Organization Dialog Box" on page 193

4. The **Navigation** menu provides creation and management functions. Select a list item and then select the desired function from the **Navigation** menu to the right of the References tab.

   Alternatively, Ctrl-click (Mac) or right-click (Windows) a list item and then select the desired function from the navigation context menu.

Menu Command	Function
Classes tab	
New	Select **New** to open the New Class dialog box. See "Creating Classes" on page 177. When a new class is created, it does not automatically become the active class.
Edit	Select a class and then select **Edit** to edit it in the Edit Class(es) dialog box. See "Setting Class Properties" on page 179. This option is not available when classes are displayed in hierarchical order and a class group header is selected.
Duplicate	Select a class and then select **Duplicate** to create a copy of it. The name of the duplicate is the same as the original class, with a number added (as in cabinets-2); if the original name ends in a number, the next available sequential number is used. This option is not available when classes are displayed in hierarchical order and a class group header is selected.
Delete	Select a class and then select **Delete** to open the Delete Class(es) dialog box. Specify what to do with the objects currently assigned to the class(es) being deleted (delete them, or reassign them to another selected class). Click **OK** to return to the Navigation palette. The program moves all objects in the deleted class(es) to the appropriate class, or deletes them, as specified. Shortcut keys for this command are Delete (Windows) and Forward Delete (Mac). This option is not available when classes are displayed in hierarchical order and a class group header is selected. The Dimension and None classes cannot be deleted. These are default classes in every drawing.
Assign to Selection	Select one or more drawing object(s) and one class; then select **Assign to Selection** to place the object(s) in the specified class
Activate	Select a class and then select **Activate** to make that class active for every design layer, viewport, and saved view without class visibility saved. (Enter is a shortcut key for this command.)
Details	Select **Details** to open the Organization dialog box to the Classes tab in Details view; double-clicking the Classes tab in the Navigation palette is a shortcut for this command

Menu Command	Function
Visibilities	Select a class and then select **Visibilities** to open the Organization dialog box to the Classes tab in Visibilities view (showing Viewport and Saved View visibilities for the class)
Select All	Choose **Select All** to select all classes
Hierarchical Display	Toggles the hierarchical display of classes on or off
Expand All	When classes are displayed in hierarchical order, displays all classes in the file
Collapse All	When classes are displayed in hierarchical order, collapses the list to display only top-level classes (those with no dash in the name) and class group headers
Design Layers tab	
New	Select **New** to open the New Design Layer dialog box. When a new layer is created, it becomes the active layer.
Edit	Select a layer and then select **Edit** to edit it in the Edit Design Layers dialog box. See "Setting Design Layer Properties" on page 165.
Duplicate	Select a design layer and then select **Duplicate** to create a copy of it. The name of the duplicate is the same as the original layer, with a number added (as in floorplan-2); if the original name ends in a number, the next available sequential number is used.
Delete	Select a design layer and then select **Delete**; when prompted, click **Yes** to confirm the deletion. Shortcut keys for this command are Delete (Windows) and Forward Delete (Mac). When a design layer is removed from the drawing, all objects in that layer are also removed.
Assign to Selection	Select one or more drawing object(s) and one layer; then select **Assign to Selection** to place the object(s) on the specified layer
Activate	Select a layer and then select **Activate** to switch to that design layer. (Enter is a shortcut key for this command.)
Update Reference	(Layer import referencing method only) This option is available if a design layer has been referenced into this file. The referenced layer name displays in italics. Position the cursor over the layer name to display a screen tip with the full layer name and the source file name; on Windows, click the disclosure arrow on the screen tip to collapse or expand it. Select a referenced layer and then select **Update Reference** to update this file with layer information from the master file. See "Workgroups and Referencing" on page 207.
Details	Select **Details** to open the Organization dialog box to the Design Layers tab in Details view; double-clicking the Design Layers tab in the Navigation palette is a shortcut for this command
Visibilities	Select a layer and then select **Visibilities** to open the Organization dialog box to the Design Layers tab in Visibilities view (showing Viewport and Saved View visibilities for the layer)
Select All	Choose **Select All** to select all design layers
Sheet Layers tab	
New	Select **New** to open the New Sheet Layer dialog box. When a new layer is created, it becomes the active layer.

Menu Command	Function
Edit	Select a sheet layer and then select **Edit** to edit it in the Edit Sheet Layers dialog box. See "Setting Sheet Layer Properties" on page 170.
Duplicate	Select a sheet layer and then select **Duplicate** to create a copy of it. The **Sheet Title** of the duplicate is the same as the original layer. The **Sheet Number** of the duplicate is the same as the original layer, with a number added (as in details-2); if the original **Sheet Number** ends in a number, the next available sequential number is used.
Delete	Select a sheet layer and then select **Delete**; when prompted, click **Yes** to confirm the deletion. Shortcut keys for this command are Delete (Windows) and Forward Delete (Mac).
Activate	Select a layer and then select **Activate** to switch to that sheet layer. (Enter is a shortcut key for this command.)
Details	Select **Details** to open the Organization dialog box to the Sheet Layers tab in Details view; double-clicking the Sheet Layers tab in the Navigation palette is a shortcut for this command
Select All	Choose **Select All** to select all sheet layers
Viewports tab	
New	Select **New** to open the Create Viewport dialog box. For sheet layer viewports, see "Creating a Sheet Layer Viewport from a Design Layer" on page 1616. For design layer viewports, see "Creating Design Layer Viewports" on page 1620.
Edit	Select a viewport and then select **Edit** to edit it in the Properties dialog box. For sheet layer viewports, see "Viewport Properties" on page 1636. For design layer viewports, see "Viewport Properties" on page 1636.
Duplicate	Select a viewport and then select **Duplicate** to create a copy of it. The name of the duplicate is the same as the original viewport, with a number added (as in details-2); if the original name ends in a number, the next available sequential number is used. The **Drawing Title** of the duplicate is the same as the original viewport. If the original viewport has a drawing label, the **Drawing Number** of the label in the duplicate viewport is the next available sequential number. The duplicate viewport is placed directly on top of the original, in the original sheet layer.
Delete	Select a viewport and then select **Delete**; when prompted, click **Yes** to confirm the deletion. Shortcut keys for this command are Delete (Windows) and Forward Delete (Mac).
Activate	Select a viewport and then select Activate to switch to that viewport's sheet layer and select that viewport (if a viewport is in a class with settings that prevent the viewport from being selected, a message displays to indicate that the active class changed to the viewport class). (Enter is a shortcut key for this command.)
Update	Select one or more sheet layer viewports, and then select **Update** to render the viewports with their foreground and background render settings. (Alternatively, select **View > Update Selected Viewports**.) When a Renderworks render mode is selected, the viewports render in the background once the geometry has been processed (see "Background Rendering" on page 1594). Icons in the Navigation palette indicate the render status of the viewports. 

Menu Command	Function
Cancel Update	When one or more selected sheet layer viewports are in the process of updating, or in the render queue, **Cancel Update** cancels the current render and any pending rendering. (Alternatively, select **View > Cancel All Viewport Updates**.)
Details	Click **Details** to open the Organization dialog box to the Viewports tab in Details view; double-clicking the Viewports tab in the Navigation palette is a shortcut for this command
Visibilities	Select a viewport and then select **Visibilities** to open the Organization dialog box to the Viewports tab in Visibilities view (showing Class and Design Layer visibilities for the viewport)
Select All	Choose **Select All** to select all viewports
Saved Views tab	
New	Select **New** to open the Save View dialog box. See "Creating Saved Views" on page 189.
Edit	Select a saved view and then select **Edit** to edit it in the Edit Saved View dialog box. See "Editing Saved Views" on page 190.
Duplicate	Select a saved view and then select **Duplicate** to create a copy of it. The name of the duplicate is the same as the original view, with a number added (as in deckview-2); if the original name ends in a number, the next available sequential number is used.
Delete	Select a saved view and then select **Delete**; when prompted, click **Yes** to confirm the deletion. Shortcut keys for this command are Delete (Windows) and Forward Delete (Mac).
Activate	Select a saved view and then select **Activate** to switch to that view. (Enter is a shortcut key for this command.)
Redefine	Select a saved view and then select **Redefine** to open the Redefine Saved Views dialog box. See "Redefining Saved Views" on page 191.
Details	Select **Details** to open the Organization dialog box to the Saved Views tab in Details view; double-clicking the Saved Views tab in the Navigation palette is a shortcut for this command
Visibilities	Select a saved view and then select **Visibilities** to open the Organization dialog box to the Saved Views tab in Visibilities view (showing Class and Design Layer visibilities for the saved view). The active class/layer and class/layer options can also be changed here.
Select All	Choose **Select All** to select all saved views
References tab	
New	Click **New** to open the Open File dialog box. For layer import references, see "Adding and Editing Layer Import References" on page 210. For design layer viewport references, see "Creating a Referenced Design Layer Viewport" on page 1622.
Edit	Select a referenced file and click **Edit** to open the Edit Reference dialog box. For layer import references, see "Adding and Editing Layer Import References" on page 210. For design layer viewport references, see "Creating a Referenced Design Layer Viewport" on page 1622.
Delete	Select a reference and click **Delete**. In the Delete Reference dialog box, specify what to do with the items in the file that are currently referenced. See "Deleting References" on page 215.
Update	Select a referenced file and click **Update** to update this document with references from the file.
Update All	Click **Update All** to update all of the references in this document.

Menu Command	Function
Open	Select a referenced file and click **Open** to open the referenced file. Referenced PDF or image files are opened with the default application for the file type. This command is not available when more than one referenced file is selected.
Details	Select **Details** to open the Organization dialog box to the References tab in Details view; double-clicking the References tab in the Navigation palette is a shortcut for this command
Select All	Choose **Select All** to select all references

Creating Classes
Creating Layers
Updating References

Workgroups and Referencing

File sharing can provide consistency and efficiency in your project files, especially in a large business where several people work on the same project. The Vectorworks program has referencing and workgroup features to make it easier to share drawing elements with your coworkers when needed.

Referencing Features

Referencing allows you to use drawing elements from another file in the current file. When a referenced item in the master file changes, the changes are reflected in the target file. Updates to target files can be performed automatically or only when manually requested.

- A “layer import” reference imports all classes and objects from a design layer in an external Vectorworks file to a design layer in the target file.
- A “design layer viewport” reference imports the contents of an external Vectorworks file to a viewport in the target file. Like any other viewport, you can then select which classes and layers to display in the viewport. (Vectorworks Design Series required)
- Resources (such as hatches, worksheets, or symbols) in other Vectorworks files can be referenced, and then used in your drawings as needed.
- In addition to Vectorworks files, you can reference DXF/DWG and DWF files (see “DXF/DWG and DWF References” on page 1728).
- If Vectorworks Design Series products are installed, you can also reference image and PDF files (see “Importing an Image File” on page 1680 and “Importing PDF Files” on page 1686).

Use the References tab of the Organization dialog box to create references to other files, and to edit or update existing references.

Workgroup Feature

When Vectorworks Design Series is installed, you can create a workgroup environment, to share one or more folders of custom content files on a network with coworkers. These files might contain content that will be used by all users (such as templates, symbols, export settings, or workspaces). Alternatively they might contain custom content specific to different projects.

This method of file sharing is best for custom content that does not change often. The content is available to multiple users without being referenced into each file, but because it is not referenced, it cannot be updated automatically.

Workgroup and Referencing Strategies

Communication among coworkers is essential when referencing and/or workgroups are in use. Discuss a strategy for how to name, maintain, and update master files. Keep each other informed of major changes—especially the deletion of a master file or resources, which can affect multiple files.

If either the file name or the location of a referenced file is changed, the reference is broken. Items referenced from that file cannot be updated until the broken reference is corrected.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Opening Referenced Files
Sharing Custom Content Using Workgroup Folders

Layer Referencing

You can reference specific layers with all of the classes and resources used in those layers, as well as any resources from a master file. Referenced design layers can be displayed in viewports. Referenced items are indicated in the Resource Browser and in the Organization dialog box by italicized names.

There are two ways to reference design layers:

- In the Vectorworks Fundamentals product, design layers are imported into the target file when they are referenced. For backward compatibility, Vectorworks Design Series products support this method; see “Adding and Editing Layer Import References” on page 210.
- In the Vectorworks Design Series products, the recommended method is to create a design layer viewport and then reference the desired design layers from the master file. See “Creating a Referenced Design Layer Viewport” on page 1622 for details about this type of reference.

Depending on the type of object referenced, you may be able to update the master file from the target file.

- Resources can be edited or renamed in the target file, and the corresponding resource in the master file is automatically updated. See “Referencing Resources” on page 213.
- Objects on layers referenced with layer import referencing are locked; they can be unlocked and edited in the target file, but the change will not appear in the master file. Moreover, when the target file’s references are updated, referenced items are over-written to reflect the master file. Therefore, any permanent change to a referenced item must be made in the master file.
- Objects on layers referenced in a design layer viewport cannot be edited directly, but there is an option to navigate to the master file, make the change, and then return to the target file to view the change.

Keep the following concepts in mind for both design layer referencing methods.

- Drawing information is shared and updated on a layer-by-layer basis.
- For layer import referencing, referenced layers should be treated as read-only layers; any changes to referenced information and any information added to referenced layers are removed the next time the referenced layer is updated.
- Referenced layer names cannot be changed.
- Resources (symbol names) in layers that are referenced from the master file take precedence over resources in the target file.
- If there is a naming conflict with pre-existing symbols or pre-existing layers in the current file, replace the symbols or rename the layers.
- Sheet layer viewports cannot directly reference layers in external master files. If you are using layer import referencing, create a reference to the master file, and select the external layers to import into the target file; then make the imported layers visible in a sheet layer viewport. If you are using design layer viewport referencing (Vectorworks Design Series required), create a design layer viewport from the master file and make the desired layers visible; then make the design layer that contains the viewport visible in a sheet layer viewport.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Referencing Options
Adding and Editing Layer Import References
Creating a Referenced Design Layer Viewport
Prioritizing Referenced Files
Updating References
Correcting Broken References
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Referencing Options

The reference settings control how manual and automatic updates to references are performed in the current file. In addition, in the Vectorworks Design Series products, you have the option to change the layer referencing method for this file from design layer viewport referencing (the default for the Vectorworks Design Series product) to layer import referencing.

To set the referencing options:

1. Select **Tools > Organization**.

   The Organization dialog box opens.

2. Select the References tab and click **Settings**.

   The Reference Settings dialog box opens. Specify the referencing settings.

Click to show/hide the parameters.

Parameter	Description
Check for out of date references every ___	Automatically checks referenced files for changes after this time interval; if any references are out of date, an alert dialog box displays to allow the target file to be updated
Use reference priority when updating resources	If the same resource exists in multiple referenced master files, deselect this option to update each referenced item from the original source, regardless of the priority order of the referenced files. This is recommended because, if you change the priority order, or if you update an individual reference (instead of all references), you could change the source file of a resource inadvertently. Select this option to update resources in the target file according to the priority order set in the References tab of the Organization dialog box (see "Prioritizing Referenced Files" on page 211). An alert message prompts you to confirm that you want to use priority updating for references.
When manually updating	Specifies what happens when the **Update** button on the References tab of the Organization dialog box is clicked: the button either updates only the selected references that are out of date, or it updates all of the selected references in the target file
Use this method to reference Vectorworks files (Vectorworks Design Series required)	Specifies which method to use to reference design layers in other Vectorworks documents. By default, Vectorworks Design Series products use design layer viewports, as described in "Creating a Referenced Design Layer Viewport" on page 1622. Select Layer import to use the referencing method described in "Adding and Editing Layer Import References" on page 210. The selected method is also used to reference DXF/DWG and DWF files, as described in "DXF/DWG and DWF Import Options" on page 1720.

3. Click **OK** to return to the Organization dialog box.

Adding and Editing Layer Import References

In the Vectorworks Design Series products, you must first select the **Layer import** referencing option for the current file, as described in “Setting the Referencing Options” on page 209. See “Creating a Referenced Design Layer Viewport” on page 1622 for information about the design layer viewport referencing option.

To add or edit references to design layers in other Vectorworks files:

1. In the active target file, select **Tools > Organization**.

   The Organization dialog box opens.

2. Select the References tab.

   Referenced files are listed in priority order, along with their current status, whether they are set to update automatically when the target file is opened, and whether they are set to save copies of the referenced items with the file.

Button	Description
New	Opens the Open File dialog box to select a new reference file
Edit	Opens the Edit Reference dialog box to change the parameters of a selected reference file
Delete	Opens the Delete Reference dialog box to delete a file from the list of referenced files, and to choose whether to keep the referenced layers and resources in the target file; see “Deleting References” on page 215
Update	Manually updates the referenced items from the selected file(s); see “Updating References” on page 212
Settings	Opens the Reference Settings dialog box to set options for updates of referenced items; see “Setting the Referencing Options” on page 209

3. To edit a current reference file, select the file and click **Edit**.

   The Edit Reference dialog box opens. Proceed to step 5.

4. To add a new reference file, click **New**.

   The Open File dialog box opens. Select the desired file, and then click **Open** to open the appropriate dialog box:

   - **Vectorworks file:** New Reference dialog box
   - **DXF/DWG or DWF file:** Import Options dialog box
   - **Image file:** Image Import Options dialog box (Vectorworks Design Series required)
   - **PDF file:** Import PDF dialog box (Vectorworks Design Series required)

   Referenced Vectorworks files must be the same version as the target file.

5. From the New Reference or Edit Reference dialog box, specify the parameters for the reference.

Click to show/hide the parameters.

Parameter	Description
Source File	Displays the path and file name of the referenced master file; if the **Edit** option was selected, click **Browse** to edit the file location

Parameter	Description
Save reference location as	Maintains either an absolute or relative file path reference from the current file to the referenced file. Use the absolute path when the location of the referenced file with respect to the current file is not going to change. Use the relative path when the files might be moved to another computer or platform; as long as the relative path between the files remains the same, the reference can be found. Both files must be saved on the same volume to select this option. The **Source File** path displays either an absolute or relative path, depending on the selection.
Layers Available	Specify the design layers to be referenced; selected layers are indicated with a check mark. Sheet layers cannot be referenced. This step is optional; a layer does not have to be selected to reference its resources. Resources in the entire referenced file are available through the **Reference** command in the Resource Browser.
Options	Select the update options
Save referenced cache to disk	Saves a copy of the referenced data with the target file. When this option is deselected, a copy of the referenced data is not saved, which means that the target file size is smaller; the referenced data is updated when the target file is opened.
Automatically update out of date reference during file open	Updates the reference each time the target file is opened; when deselected, the reference is updated only when **Update** is clicked from the References tab of the Organization dialog box
Update class definitions	Updates class definitions along with the referenced objects that use those classes
Ignore source user origin	Ignores the user origin in the master file; referenced objects will not change location in the target file if the user origin changes in the master file
Create layer link(s) on layer	Creates a layer link for each newly referenced layer on the specified design layer; select the design layer from the list or select New Layer to specify a new design layer, set to the same scale as the first selected layer to reference

Referenced layers can be used in a sheet layer viewport. Because sheet layer viewports cannot directly reference design layers in other files, reference the layers and then create a sheet layer viewport that shows the referenced layers.

6. Click **OK** to return to the Organization dialog box.

   The selected layers are added to the target file, along with any new layers and layer links.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Referencing Options
Prioritizing Referenced Files
Updating References
Correcting Broken References
Referencing Resources
Deleting References
Opening Referenced Files
Layer Referencing
Creating a Referenced Design Layer Viewport

Prioritizing Referenced Files

During a reference update, the program can search the referenced files in the order in which they are listed on the References tab, and then update the target file with the first instance of each referenced resource that is found.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

If the same resource exists in multiple referenced master files, it is recommended that you disable this feature. Otherwise, if you change the priority order, or if you update an individual reference, you could change the source file of a resource inadvertently. To prevent this from happening, deselect the option to **Use reference priority when updating resources** in the Reference Settings dialog box (see "Setting the Referencing Options" on page 209).

To set the priority of referenced files:

1. Select **Tools > Organization** to open the Organization dialog box.
2. From the References tab, click **Settings** to open the Reference Settings dialog box, and ensure that the option to **Use reference priority when updating resources** is selected.
3. From the References tab, click the number in its # column to select the master reference file(s) to be moved; drag the item up or down the list to the new priority position. The first file listed on the References tab has the highest priority.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding and Editing Layer Import References
Creating a Referenced Design Layer Viewport

Updating References

Referenced design layers and resources are either updated automatically when the target file is opened, or they are updated manually upon command. These preferences are set in either the New Reference or the Edit Reference dialog box. If the file is set to update automatically, manual updates can still be performed at any time.

Before an update, correct any broken references as described in "Correcting Broken References" on page 213.

To update references manually:

1. Select **Tools > Organization** to open the Organization dialog box.
2. From the References tab, select the master file(s) that contain the referenced items to be updated.

   To update all references in the target file, select all of the files.
3. Click **Update** to update the referenced items in the target file from the selected files.
   - If priority updating is disabled (recommended), referenced items are always updated from the original file.
   - If priority updating is enabled, referenced items are updated according to the priority order on the References tab.

   For more information about priority updating, see "Setting the Referencing Options" on page 209, and "Prioritizing Referenced Files" on page 211.
4. If a referenced item is missing, the Referenced Resource not Found dialog box opens. Select an action option.

Click to show/hide the parameters.

Parameter	Description
Break the reference	Breaks the reference with the master file, leaving the resource in the target file
Delete the resource	Deletes all occurrences of the resource in the target file (from both the drawing and the Resource Browser)
Replace resource with	To replace the missing resource with another item from the master file, select this option, and then select the replacement item from the list of available resources

5. Click **OK To All** to perform the same action for all missing resources. To select an action for each missing resource individually, click **OK**; the dialog box redisplays for each missing item.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Correcting Broken References

Correcting Broken References

If a reference has a status of "Broken link" in the References tab, the program cannot locate the master file. Items referenced from that file cannot be updated until the broken reference is corrected.

To correct broken references:

1. Select **Tools > Organization** to open the Organization dialog box.
2. From the References tab, select one or more master files that have broken references, and click **Update**.
3. An alert prompts you to locate the master file. If you selected multiple references, select **Look for subsequent broken references in all of the folders manually located** to have the program search the same location for all of them.

   If a file is set to update automatically when it is opened, and one of its references is broken, this alert displays when you open the file.
4. Click **Yes** to open the Open Current-Version Drawing dialog box.
5. Locate the master file and click **Open**.

   On the References tab, the status of the broken reference changes from Broken link to Up to date or Out of date. If you selected the option to look for subsequent references in the same location, those references are corrected also.
6. Click **OK** to save the new name and/or location for the references.

Updating References

Referencing Resources

The resources of files listed in the References tab of the Organization dialog box, as well as the resources in any Vectorworks file of the same version number, can be referenced through the Resource Browser.

To reference the resources of a master file:

1. To display the resources of the master file in the Resource Browser, either use the **Browse a Document** command on the **Files** menu, or use the **Files** list to access the master file if it is in a resource library (see "Accessing Existing Resources" on page 229).
2. Right-click (Windows) or Ctrl-click (Mac) on the desired resource, and select **Reference** from the context menu.
3. If the resource is a symbol or plug-in object, the Reference Symbol(s) dialog box opens. Specify the location of the referenced symbol and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Preserve folder hierarchy	Maintains the same folder structure the referenced symbol had in its source file
Select destination folder	Adds the referenced symbol to the selected folder
Folder list	Displays the target file's symbol folders; the selected folder becomes the destination for the referenced symbol
New Folder	Creates a new folder within the selected folder; specify the folder name and click **OK** to create the symbol folder

4. If the master file has never been referenced in the current file, the New Reference dialog box opens. Specify the reference parameters and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Source File	Displays the path and file name of the referenced resource file
Save reference location as	Maintains either an absolute or relative file path reference from the current file to the referenced resource's file. Use the absolute path when the location of the referenced file with respect to the current file is not going to change. Use the relative path when the files might be moved to another computer or platform; as long as the relative path between the files remains the same, the reference can be found. Both files must be saved on the save volume to select this option. The **Source File** path displays either an absolute or relative path, depending on the selection.
Save referenced cache to disk	Saves a copy of the referenced resource with the target file. When this option is deselected, a copy of the referenced resource is not saved, which means that the target file size is smaller; the referenced resource is refreshed when the target file is opened.
Automatically update out of date reference during file open	Updates the reference each time the target file is opened; when deselected, the reference is updated only when **Update** is clicked from the References tab of the Organization dialog box
Update class definitions	Updates class definitions along with the referenced resources that use those classes

5. The resource is added to the target file. Referenced resources display with italicized names in the Resource Browser.

   The following rules apply to referenced resources.

Situation	Description
Editing a referenced resource	A referenced resource can be edited within the target file, provided the source file is not opened by another user, and the reference is not broken or the resource out of date. Changes to the referenced resource are automatically exported to the source file; while in edit mode, the source file is locked. The following resources can be edited within the referenced file: gradients, hatches, record formats, symbol definitions, plant symbols (Vectorworks Landmark required), sketch styles and wall styles (Vectorworks Design Series required), Renderworks backgrounds and textures (Renderworks required).
Resource name conflict	If there is a name conflict between a referenced object and another object when creating a reference, an alert displays and you have the option to replace the object in the current document. (If the other object is also a referenced object, it cannot be replaced and the update fails.)
Change user origin of master file	If the user origin of a master file is changed, the location of the referenced resources do not change, if **Ignore source user origin** is selected in the Edit Reference dialog box
Change master file classes	Changes to the class of a referenced resource are only reflected in the target file after an update if **Update class definitions** is selected in the New Reference dialog box

Situation	Description
Resource deleted in master file	If a referenced resource is deleted in a master file and the resource does not exist in any other master file included in an update, when the reference to the master file is updated, an alert displays. Click **OK** to confirm that the resource is an "orphan," or click **OK to All** to hide all alerts about orphaned resources during the current update. Orphaned resources are no longer referenced.

To break a reference, right-click (Windows) or Ctrl-click (Mac) on the desired resource, and select **Break Reference** from the context menu. The resource remains in the target file, but it is no longer referenced to the master file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding and Editing Layer Import References
Creating a Referenced Design Layer Viewport

Deleting References

You can stop referencing a file, and choose whether to keep the referenced layers and resources in the target file.

To delete a reference to a file:

1. Select **Tools > Organization** to open the Organization dialog box.
2. Select the file to stop referencing, and then click **Delete**.

   The Delete Reference dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Reference to	Displays the master file name and path
Referenced Layers	Select an option for layers that are currently referenced in the target file from the master file
Keep in current document	Keeps the layers in the target file, but removes the reference
Delete	Removes referenced layers from the target file, including layer links and any objects on the layer
Referenced Resources	Select an option for resources that are currently referenced in the target file from the master file
Keep in current document	Keeps the resources in the target file, but removes the reference
Delete if unused	Deletes unused referenced resources; keeps resources that have been placed in the file, but removes the reference

3. Specify what to do with the referenced layers and resources, and click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

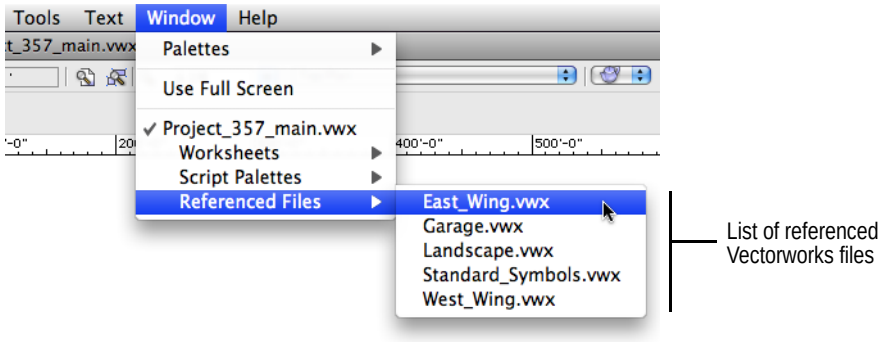
Layer Referencing

Opening Referenced Files

The Vectorworks program provides easy access to the master files that a target file references.

To open a referenced file:

1. Click on **Window**. The bottom portion of the menu provides a submenu for each Vectorworks file that is currently open.
2. From the submenu for the target file, select **Referenced Files** to display the Vectorworks files that the target file references. Though DXF/DWG, DWF, PDF, and image files can be referenced, they do not appear on the menu.
3. Select a referenced file to open it.



D Sharing Custom Content Using Workgroup Folders

When Vectorworks Design Series is installed, you can create a workgroup environment, to share one or more folders of custom content files on a network with coworkers. These files might contain content that will be used by all users (such as templates, symbols, export settings, or workspaces). Alternatively they might contain custom content specific to different projects. Each user who needs access to those files can then designate that shared network folder as a workgroup folder in his or her Vectorworks Preferences, so that the custom content will be available in the Vectorworks program.

You can manually add files to the workgroup Libraries folder, to access custom resource libraries such as symbols and hatches from the Files list in the Resource Browser. You can also have custom resources display as default content in dialog boxes and palettes. See “Resource Libraries” on page 219 for more information. For details about where to put files and how to name them, see “Sharing Custom Files with a Workgroup” on page 217.

Designating Workgroup Folders

To designate workgroup and project folders:

1. Select **Tools > Options > Vectorworks Preferences**, and then click the User Folders tab.

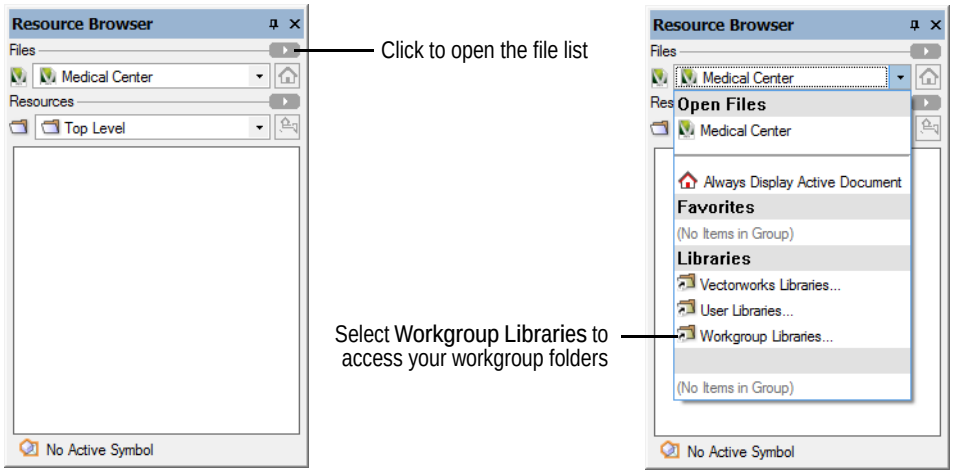
Click to show/hide the parameters.

Parameter	Description
User Data and Preferences Folder	Specifies the folder that contains Vectorworks preferences, log files, workspaces, and any personal content you create. This might be a folder on the local computer, or on a USB drive or network drive; this allows you to run the Vectorworks program from any computer. For more information about user folders, see “User Folders Preferences” on page 57.
Choose	Click **Choose** to change the user data folder. The Vectorworks program must be restarted if you change the location of the user data. See “User Data and Preferences Folder” on page 58 for details.

Parameter	Description
Explore (Windows) or Reveal in Finder (Mac)	To look at the contents of the current folder, click **Explore** (to open Windows Explorer) or click **Reveal in Finder** (to open Mac Finder)
Workgroup and Project Folders	Specifies the folders where the Vectorworks program will look for additional content, such as office standard files, shared project files, and workgroup reference files. This might be a shared folder on a network drive, or a project-specific folder on the local computer. Folders are searched in order from the top of the list to bottom; if multiple files with the same name are found, only the first version of the file is shown in the program. To change the position of a folder, click in the # column and drag the folder up or down in the list.
Add	Opens a dialog box to select a folder to add to the list
Remove	Deletes the selected folder from the list
Explore (Windows) or Reveal in Finder (Mac)	Displays the contents of the selected folder in either Windows Explorer or Mac Finder

2. Enter the workgroup folder(s) as needed, and click **OK** to save the changes.

   To easily access a workgroup folder, use the file list in the Resource Browser. See "Accessing Existing Resources" on page 229 for more information.



Accessing workgroup folders from the Resource Browser

Sharing Custom Files with a Workgroup

To share custom content with a workgroup, create a main workgroup folder in a shared location. Within the folder, create a subfolder that matches the standard Vectorworks program subfolder for that type of data (Libraries, Plug-Ins, Settings, or Workspaces). If necessary, create additional subfolders within those folders (for example, Libraries\Defaults\Templates or Libraries\Defaults\Space\Space - Space Name).

Do not modify the Plant Database subfolder, if present. Place custom plant symbols in one of the plant library subfolders (in the Libraries\Defaults folder).

To access the shared content, another Vectorworks user simply designates the main folder as a workgroup folder in his or her Vectorworks Preferences. Here are some examples of possible workgroup folder content:

- Master files from which design layers, classes, and resources can be referenced
- Plug-ins

- Templates
- Resource libraries (for gradients, hatches, line types, record formats, Renderworks textures, slab styles, symbols, text styles, tiles, VectorScripts, wall styles, and worksheets)
- Space object lists (space names, occupant organizations, and room finishes)

Sharing Custom Resource Libraries

When the Vectorworks program presents content in a palette or dialog box, it includes content from the user and workgroup folders as well as the content it ships with. For example, when you select **Modify > Hatch** to edit a hatch, the Hatches dialog box shows default hatches from the Vectorworks folder (where the software is installed), your user folder, and any workgroup folders you have designated.

If the same file name exists in more than one folder, only the content from one of the files is shown in the Vectorworks program, according to the following priority: user folder, workgroup folder, application folder. For example, if the following files exist, only hatches from the file in the user folder show in the hatch list in the Attributes palette in the Vectorworks program.

- [User]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx
- [Workgroup]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx
- [Vectorworks]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx

Therefore, the name of your custom library file affects content availability in the Vectorworks program.

- To totally supersede the Vectorworks default hatches, use the same name as the hatch file in the Vectorworks program folder (“Hatches_Default.vwx”).
- To make your custom hatches available along with the Vectorworks default hatches, use a unique name, such as “Custom_Hatches.”

To share a custom library, create a library file and put the file on the network in a workgroup folder. The folder structure must be the same as that of the Vectorworks default content. For example, to add a custom hatch library, place the file in your workgroup default hatch folder: [Workgroup]\Libraries\Defaults\Attributes - Hatches.

If the same resource name exists in two default files (that have different names), the resource list in Vectorworks shows both resources, and includes the file name. For example, on the Attributes palette you might see two hatches named “Brick (Custom_Hatches.vwx)” and “Brick (Hatches_Default.vwx).” You can use both hatches in a drawing, but after you import the first one, you are prompted to either replace the first one, or rename the second one.

> To make the custom library resources available from the Resource Browser only (not as defaults in dialog boxes and palettes), place the file in a Favorites folder in a workgroup folder. See “Using Favorites Files” on page 230. This method of sharing resources is more flexible, because users can choose either to import the resource, or reference it (see “Referencing Resources” on page 213).

The Resource Browser

Resource Libraries

Vectorworks software comes with hundreds of libraries, which can be accessed through the Resource Browser (see “Accessing Existing Resources” on page 229). Default content (which is available at the time of use) is automatically installed with the Vectorworks software. All other content, including industry-specific content, is optionally added during the installation process (Internet access required). If the additional content was not added during installation, it can be added later. Select **Help > Download Content** to download it via the Vectorworks Package Manager.

Excluding standards and templates (which are installed based on the drawing unit selected during installation), all libraries for the Vectorworks Fundamentals and Renderworks products are universal, and are installed regardless of the drawing unit selected during installation.

Libraries are organized by professional discipline and subject. The Vectorworks Design Series products contain additional libraries. Even more content is available exclusively for Vectorworks Service Select subscribers.

For a list of libraries and the product(s) to which they belong, see the Libraries-Index and Product Matrix.pdf file located in the [Vectorworks]\Libraries folder.

All of the Vectorworks interior and building objects (symbols) specifically developed for Architect and Designer have ifc data records attached. This includes the symbols libraries available in Vectorworks Architect and Designer products, as well as libraries available in our subscription service, Vectorworks Service Select.

A subset of content (some of which are resources available in libraries) is also available by default at the point of use. The ability to use default content is a preference on the Session tab of the Vectorworks preferences, which can be disabled if default content use is not desired; see “Session Preferences” on page 52.

Default content is available from a variety of dialog boxes and also from the Attributes and Object Info palettes. Once default content is selected for use, it is automatically imported into the current file and if it is a resource (such as a hatch or gradient) it also displays in the Resource Browser.

Creating Custom Default Content Libraries

To make your own custom content available as default content in the Vectorworks program, use any of the following methods.

- Import custom content into a default content file
- Add a custom content file to a default content folder
- Create a custom content file and place an alias of (Mac) or shortcut to (Windows) that file in a default content folder

Files that contain custom content can be placed in different locations, depending on who needs access to the content. The folder structure must be the same as that of the Vectorworks default content, and if a subfolder exists, the file must be placed in a subfolder. For example, Vectorworks default hatches are in a file in the folder [Vectorworks]\Libraries\Defaults\Attributes - Hatches. To make your library of custom hatches available in Vectorworks, place a file that contains the hatches in either your user folder or a workgroup folder.

- To create default content for yourself only, put the file in the appropriate subfolder within [User]\Libraries\Defaults (where [User] is the user data folder specified in your Vectorworks preferences). For example, place a hatches file in [User]\Libraries\Defaults\Attributes - Hatches. See “User Folders Preferences” on page 57 for more information.
- If Vectorworks Design Series products are installed, you can create default content that is project-specific, or that is shared with a workgroup. To do so, put the file on the network in the appropriate subfolder within [Workgroup]\Libraries\Defaults (where [Workgroup] is a workgroup folder specified in your Vectorworks preferences). For example, place a hatches file in [Workgroup]\Libraries\Defaults\Attributes - Hatches). When coworkers set up this workgroup folder in their preferences, they also have access to the default content. See “Sharing Custom Files with a Workgroup” on page 217 for more information.

The name of your custom library file also affects resource availability in the Vectorworks program.

- To totally supersede the Vectorworks default hatches, use the same name as the hatch file in the Vectorworks program folder (“Hatches_Default.vwx”).
- To make your custom default hatches available along with the Vectorworks default hatches, use a unique name, such as “Custom_Hatches.vwx.”

If a file name is repeated in any of the folders, only the content from one file is shown, according to the following precedence: user folder, workgroup folder, application folder. For example, if the following files exist, only hatches from the file in the user folder show in the hatch list in the Attributes palette in the Vectorworks program.

- [User]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx
- [Workgroup]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx
- [Vectorworks]\Libraries\Defaults\Attributes - Hatches\Hatches_Default.vwx

If a custom library file contains multiple types of content, the program only displays the content type for the folder that contains the custom file (or an alias or shortcut to the file). For example, if a file contains both wall hatches and textures, and the file is in the Libraries\Defaults\Walls - Textures folder, only the wall textures are displayed as defaults in the Vectorworks program.

> To make the custom library resources available from the Resource Browser only (not as defaults in dialog boxes and palettes), place the file in a Favorites folder in either your user folder or a workgroup folder. See “Using Favorites Files” on page 230.

Organizing Content Display

In dialog boxes and palettes, the custom and default content are displayed below the content in the current file; each list is in ascending alphabetical order.

To display the custom or current file’s content first in a selection list, implement a naming standard that places the desired content toward the top of the list alphabetically. To view only the current file’s content in a selection list, disable the **Display default content** Vectorworks preference.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Resource Libraries
Using Resources from Open Files, Favorites, and Libraries
Using the Resource Browser

Creating Resource Libraries

Resource libraries are simply files that contain specific resources to make it faster and easier to access the resources while drawing in the Vectorworks program. Libraries do not increase the size of the current file or consume significant amounts of memory. Save similar resources in the same library file to make it easier to search for them.

Once created, resource libraries can be accessed from the Resource Browser as a Favorites file, or located using the User Libraries or Workgroup Libraries options on the Files list. Resource libraries can also be used as custom default content.

To create a resource library file:

1. Select **File > New**.
2. Import or create the resources to be included in the library.
3. Select **File > Save**.

   The Save Vectorworks Drawing dialog box opens.
4. Select the location for saving the file.
5. Enter a name for the file in the **Name** field.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

6. Click **Save**.
7. Place the library file in the appropriate location, depending on how you want to access it:

Access Method	Library File Location	Reference
From the **Files** list in the Resource Browser, select the library file from the Favorites section, and then select a resource from the resource display window	Favorites folder in your user folder: [User]/Libraries/Favorites/[folder name]	"Using Favorites Files" on page 230
From the **Files** list in the Resource Browser, select the User Libraries option, select the library file, and then select the resource from the resource display window	Libraries folder in your user folder: [User]/Libraries/[folder name]	"User Folders Preferences" on page 57
From the **Files** list in the Resource Browser, select the Workgroup Libraries option, select the library file, and then select the resource from the resource display window	Libraries folder in a network folder designated as your workgroup folder (Vectorworks Design Series required): [Workgroup]/Libraries/[folder name]	"Sharing Custom Files with a Workgroup" on page 217
Available from anywhere default content resources are available, such as the hatches list in the Attributes palette	Defaults folder in your user folder, or in a network folder designated as your workgroup folder (Vectorworks Design Series required): [User]/Libraries/Defaults/[standard folder name] [Workgroup]/Libraries/Defaults/[standard folder name]	"Creating Custom Default Content Libraries" on page 219

Organizing Content Display
Using Resources from Open Files, Favorites, and Libraries
Using the Resource Browser

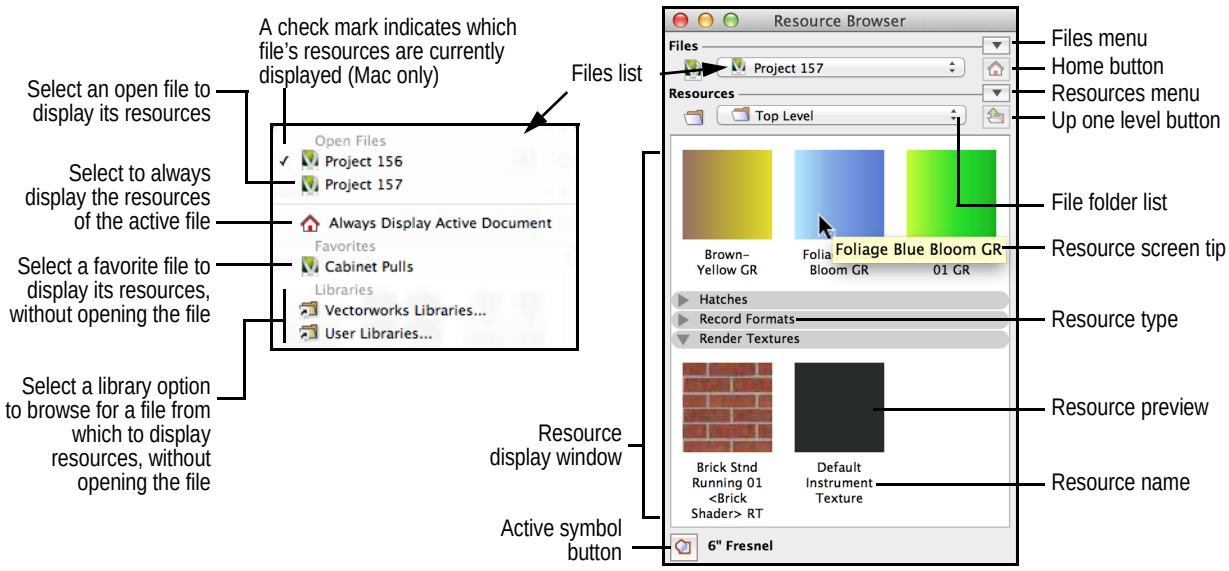
Using the Resource Browser

The Resource Browser is the central repository for all Vectorworks resources. Use it to create, apply, and manage resources in the current file, and to access resources in other files.

To use the Resource Browser:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.



Feature	Description
Files menu	Lists menu commands that are used to create and manage a set of favorite resource files and to browse the resources in an existing document (see "Quick Resource Browsing" on page 229)
Files list	Specifies the file from which to display resources in the Resource Browser. Select a file from the Open Files or Favorites sections of the list, or select an option from the Libraries section of the list and browse to locate the file that contains the resources you need. Select **Always Display Active Document** to automatically display the resources in the active file whenever you switch from one open file to another. See "Using Resources from Open Files, Favorites, and Libraries" on page 229.
Home button	Displays the resources of the currently active file
Resources menu	Lists menu commands that are used to work with resources; see the following: • "Viewing Resources" on page 223 • "Creating New Resources" on page 228 • "Finding Resources" on page 232 • "Working with Resources" on page 225
Up one level button	Moves one level up the symbol folder or script palette folder hierarchy
File folder list	Lists the file's resource folders, if there are symbol folders or script palette folders in the file. Select a folder to display the folder's resources in the resource display window. To return to a view of the entire resource hierarchy, select Top Level. All resource types and folders that are currently selected to be shown display in the resource display window.
Resource type	In Thumbnails mode, displays resources divided into categories by type; the types are displayed in headings that can be shown and expanded, or collapsed and hidden (see "Hiding and Showing Resources" on page 225). In List mode, click the column header to sort the columns by resource name or by resource type.
Resource screen tip	Shows the full resource name as a screen tip when you position the cursor over a resource; for referenced resources, the screen tip also shows the source file name
Resource preview	Displays resources with a preview image, or lists the resources with a representational icon

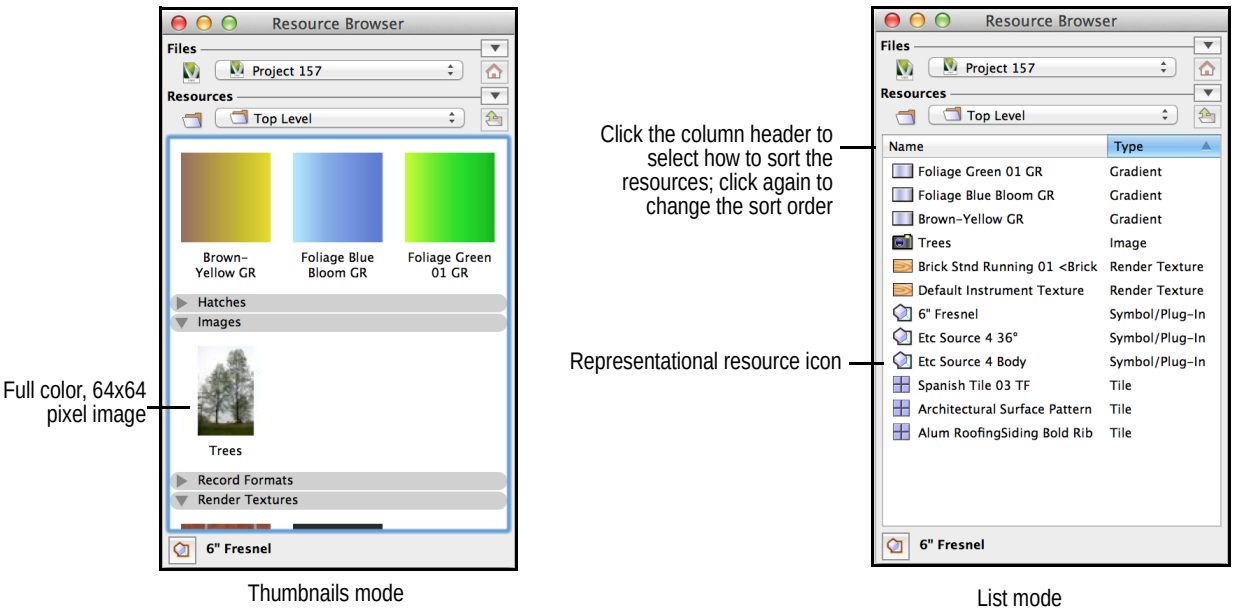
Feature	Description
Resource name	Shows the resource name; a name in italics indicates a referenced resource. Type the first letter of a resource to quickly display resource names beginning with that letter. The color of a symbol (black, red, or blue) indicates the symbol type (see “Symbol Types” on page 237).
Resource display window	Displays the resources from the file shown in the Files list, and the symbol folder shown in the File Folder list. Select which resource types to display from the **Resources** menu (see “Hiding and Showing Resources” on page 225).
Active symbol button	Displays the currently active symbol in the resource display window

2. By default, the resources from the currently active document display. Locate and select a resource to use from this file, or use the **Files** list to view resources from other files. If the resource you need does not exist, you can create one.
3. Once a resource is selected, you can use it in the drawing, or select another action from the **Resources** menu, such as Edit or Export to another file. See “Working with Resources” on page 225 for more details.

   Click here for a video tip about this topic (Internet access required).

Viewing Resources

Resources can be viewed in either Thumbnails (default) or List mode.



In Thumbnails mode, resources are displayed in 64x64 pixel full color. Gradient fills, hatch fills, image fills, line types, symbols, plug-in objects, text styles, and tile fills are previewed with an actual image; other resources are previewed with a representational icon. For symbols, the thumbnail view and render mode can be set from the **Resources** menu, and when editing symbols, the thumbnail view can be set to match the editing view. Long resource names display on up

to five lines, and the number of characters displayed depends on the system font selected. In List mode, resources are displayed as 16x16 pixel representational icons.

Resource	Thumbnail Icon	List Icon
Gradient Fill (see “Using Gradient Fills” on page 1115)		
Hatch Fill (see “Using Hatch Fills” on page 1104)		
Image Fill (see “Using Image Fills” on page 1119)		
Line Type (see “Using Line Types” on page 1128)		
Record Format (see “Record Formats” on page 262)		
Referenced Resource (see “Referencing Resources” on page 213)	Resource name displays in italics	
Renderworks Background (Renderworks required; see “Creating Layer Backgrounds” on page 1522)		
Renderworks Texture (Renderworks required; see “Creating Textures” on page 1505)		
Renderworks Prop Texture (Renderworks required); see “Creating Image Prop Objects” on page 1520		
Renderworks Style (Renderworks required); see “Creating Renderworks Styles” on page 1596		
Sketch Style (Vectorworks Design Series required; see “Sketch Rendering” on page 1497)		
Slab Style (Vectorworks Architect required; see “Creating Slabs” on page 481)		
2D Symbols and Plug-in Objects (see “Symbols” on page 237)		
3D Symbols and Plug-in Objects (see “Symbols” on page 237)		
Hybrid Symbols and Plug-in Objects (see “Symbols” on page 237)		
Symbol Folder (see “Managing Symbols” on page 251)		
Text Style (see “Using Text Styles” on page 393)		
Tile Fill (see “Using Tile Fills” on page 1111)		
Script (see “Using Scripts” on page 1771)		

Resource	Thumbnail Icon	List Icon
Script Palette (see “Using Scripts” on page 1771)		
Wall Style (Vectorworks Architect or Landmark required)		
Curtain Wall Style (Vectorworks Architect or Landmark required)		
Worksheet (see “Creating Worksheets” on page 1315)		

To change the view mode:

1. In the Resource Browser, select **Resources > View As**.
2. From the list, select **Thumbnails** or **List**.

   Select **Window > Palettes > Save Palette Positions** to save selected views across Vectorworks sessions. To revert to settings established when the custom workspace was created (in the User Data and Preferences folder), click **Reset Saved Settings** from the Session tab of Vectorworks preferences.

Hiding and Showing Resources

When there are a large number of resource types, it may be useful to hide unused resource types from view.

To hide or show resource types:

1. In the Resource Browser, select **Resources > Show Object Types**.
2. Select the resource type to hide or show. A currently visible resource type is indicated by a check mark next to its name. (In List view, hidden resources are not shown in the resource display window.)

In Thumbnails mode, click on the disclosure arrow next to the resource type name in the Resource Browser to hide or display resources. Alternatively, double-click on the resource name bar to hide or display resources.

Select **Window > Palettes > Save Palette Positions** to save the current show/hide status across Vectorworks sessions. To revert to settings established when the custom workspace was created (in the User Data and Preferences folder), click **Reset Saved Settings** from the Session tab of Vectorworks preferences.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using the Resource Browser
Session Preferences

Working with Resources

Available commands are determined by the resource selected.

To use a resource:

1. Select the resource to use in the Resource Browser.
2. Select the desired command from the **Resources** menu.

   Right-click (Windows) or Ctrl-click (Mac) on a resource to access a context menu. This menu lists the associated commands from the **Resources** menu for the selected resource.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Command	Description	Shortcut Key Equivalent
Apply	Applies the selected resource to the selected drawing object(s)	Double-click
Attach Record	Opens the Attach Record dialog box, to attach a record to the selected plug-in object, or to the symbol definition (changing all symbol instances, existing and future); see “Attaching Record Formats to a Symbol Definition” on page 264	Not applicable
Delete	Deletes the selected resource from the Resource Browser	Delete key
Duplicate	Opens the Assign Name dialog box; enter a name and click **OK** to create a duplicate of the selected resource	Not applicable
Edit	Opens an edit dialog box for the selected resource. If the resource is referenced, an alert prompts you to verify that you want to edit the resource before the edit dialog box opens. Edits made in the current document are also saved automatically to the source document, which may affect references to this resource in other files.	Ctrl-double-click (Windows) Option-double-click (Mac)
Edit 2D Component	Edits the 2D component of the symbol definition (see “Object Editing Mode” on page 1004)	Ctrl-double-click (Windows) Option-double-click (Mac)
Edit 3D Component	Edits the 3D component of the symbol definition (see “Object Editing Mode” on page 1004)	Ctrl-double-click (Windows) Option-double-click (Mac)
Edit 3D Wall Hole Component	Edits the 3D wall hole component of the symbol definition (see “Adding a 3D Wall Hole Component to a Symbol Definition” on page 260)	Ctrl-double-click (Windows) Option-double-click (Mac)
Edit Definition (Vectorworks Landmark required)	Edits the definition of the plant symbol as described in “Creating Plant Definitions” on page 787	Ctrl-double-click (Windows) Option-double-click (Mac)
Edit Symbol Options	Opens the Symbol Options dialog box to specify wall insertion options and other symbol parameters. See “Editing the Symbol Options of Multiple Symbols” on page 250.	Ctrl-double-click (Windows) Option-double-click (Mac)
Enter	Opens the selected symbol folder or script palette to display the symbols or scripts inside	Double-click
Export (Vectorworks Design Series required)	Opens the Export Resource dialog box to export the selected resource to another file. See “Exporting Custom Resources” on page 234.	Not applicable

Command	Description	Shortcut Key Equivalent
Extract Image(s)	Opens the Select Folder (Windows) or Export Image File (Mac) dialog box to save a .png file extracted from the selected image or texture resource To extract images from all image-based textures, Renderworks backgrounds, and image resources in the current document, execute this command with no resources selected.	Not applicable
IFC Data (Vectorworks Architect or Landmark required)	Attaches IFC data to the selected symbol definition, changing all symbol instances, existing and future; see “Assigning IFC Data to Objects” on page 1746	Not applicable
Make Active	Makes the selected symbol or plug-in active, and activates the **Symbol Insertion** tool so that the symbol or plug-in can be inserted	Double-click
Move	Opens the Move Symbol dialog box to move the selected symbol or plug-in object to another folder	Not applicable
New [resource]	Opens a dialog box to create a new resource of the same type as the one selected (for example, a new gradient, record format, or worksheet)	Not applicable
Open	When a worksheet is selected, opens the worksheet on screen for edits; when a script palette is selected, opens the palette	Ctrl-double-click (Windows) Option-double-click (Mac)
Rename	Opens the Assign Name dialog box; enter a name and click **OK** to rename the selected resource. If the resource is referenced, an alert prompts you to verify that you want to rename the resource before the Assign Name dialog box opens. The name change made in the current document is also saved automatically to the source document, which may break references to this resource in other files. Referenced symbols, gradients, record formats, hatches, tiles, Renderworks backgrounds, textures, sketch styles, wall styles, text styles, slab styles, plants, and image resources can be renamed.	Ctrl-double-click (Windows) Option-double-click (Mac) (Image fills only)
Import	When a resource from a different file is selected, imports the resource into the current file	Not applicable
Reference	When a resource from a different file is selected, creates a reference to it in the current file. See “Referencing Resources” on page 213.	Not applicable
Break Reference	When a referenced resource is selected, breaks the link between the current file and the master file; the resource remains in the current file but is no longer referenced	Not applicable
Run	Runs the selected script. Text documents that contain scripts must be run using the **Tools > Plug-ins > Run Script** command.	Double-click

Command	Description	Shortcut Key Equivalent
Worksheet On Drawing	Places the selected worksheet on the drawing for display and printing	Double-click
Set Thumbnail View	Applies a view to the thumbnail preview of the selected symbol(s), or to all symbols within folders and subfolders if a symbol folder is selected, or to all symbols in the file if nothing is selected	Not applicable
Set Thumbnail Render Mode	Applies a render mode to the thumbnail preview of the selected symbol(s), or to all symbols within folders and subfolders if a symbol is selected, or to all symbols in the file if nothing is selected	Not applicable

Image resources can be compressed to reduce the Vectorworks file size. See "Compressing Images" on page 1822 for more information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Accessing Existing Resources
Using the Resource Browser

Creating New Resources

There are various methods to create new resources in a drawing. Most new resources are created through the Resource Browser, as described here. Some types of resources can also be created from menu commands elsewhere in the Vectorworks program, such as scripts and text styles. Symbols display in the Resource Browser, but they can only be created with the **Modify > Create Symbol** command.

To create a new resource:

1. In the Resource Browser, select **Resources > New Resource** to display the list of new resource types.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) in the Resource Browser to open the menu.

   Double-click in an unused area of the resource display area to open the Create Resource dialog box. The dialog box contains the list of new resource types.

2. Select the resource type to create.

   The resource-specific dialog box opens.

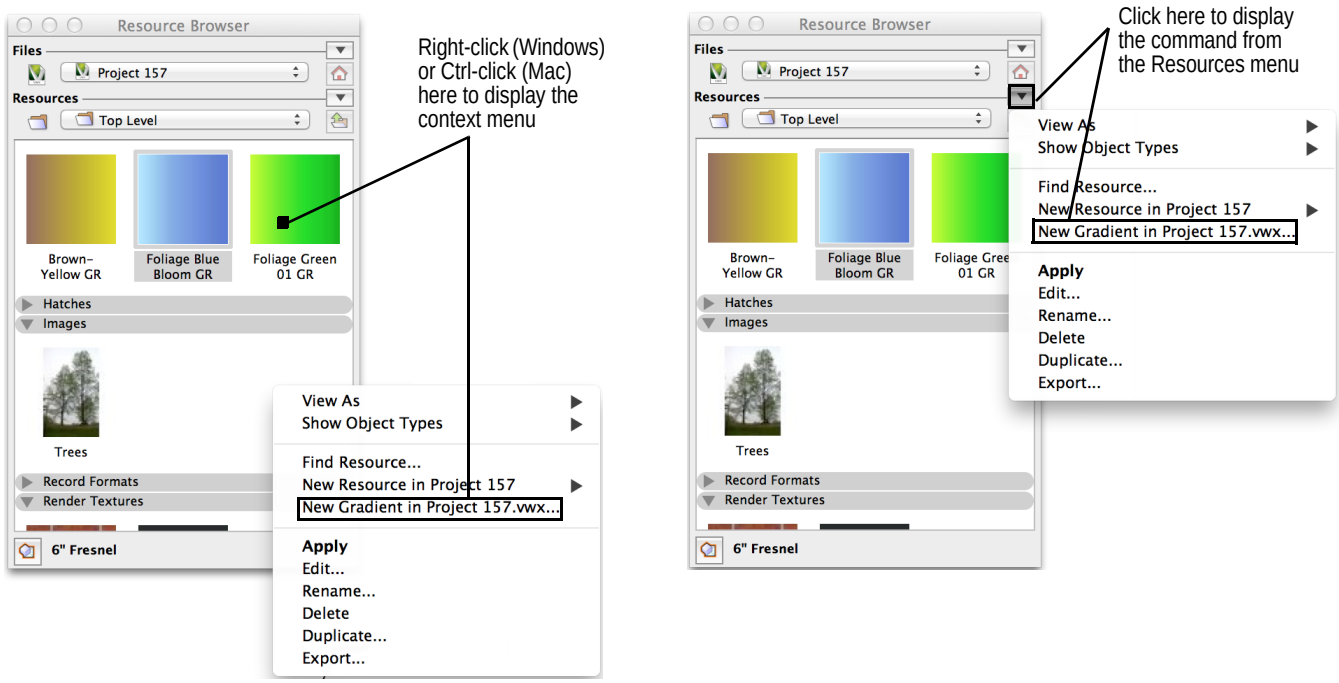
3. Follow the specific creation procedures for that resource.

   Once created, the resource displays in the Resource Browser.

Quick Resource Creation

When you right-click (Windows) or Ctrl-click (Mac) in an area of the Resource Browser that is associated with an existing resource type, the context menu contains an additional option for quickly creating another resource of the same type. For example, right-click or Ctrl-click in the Hatches section and the **New Hatch in (document)** option displays. The context menu that displays varies slightly, depending on whether you click a blank area of the Resource Browser, or directly on a resource.

Also, immediately after a new resource is created, the **Resources** menu contains an additional option for quickly creating another resource of the same type.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Finding Resources
Identifying Duplicate Resources
Working with Resources
Using the Resource Browser
Creating Resource Libraries

Accessing Existing Resources

While new resources are automatically added to the Resource Browser upon creation, resources that exist in other files must be imported or referenced into the current file through the Resource Browser palette. Default content is automatically imported into the current file at the point of use, and if it is a resource (such as a hatch or gradient) also displays in the Resource Browser; see “Resource Libraries” on page 219.

Quick Resource Browsing

Resources in another file can be quickly browsed to locate resources and to determine whether that file should be added as a favorite.

To view the resources available in another file that is not yet a favorite:

1. In the Resource Browser, select **Files > Browse a Document**.

   The standard file open dialog box opens.

2. Select the file to browse, and then click **Open**.

   The resources in the selected file temporarily display in the Resource Browser. A resource from that file can be imported into the current file, but the browsed file is not saved as a favorite.

Using Resources from Open Files, Favorites, and Libraries

The **Files** list displays all currently open files and favorites. In addition, the **Files** list provides easy access to resource libraries—both the hundreds of libraries delivered with the Vectorworks software, and any other custom libraries that you create.

- Vectorworks Libraries accesses the libraries installed with the Vectorworks program.

- User Libraries accesses the libraries placed in your User Data and Preferences folder. See “User Folders Preferences” on page 57 for more information about user folders.
- Workgroup Libraries accesses the libraries placed in your Workgroup and Project folders (Vectorworks Design Series required). See “Sharing Custom Files with a Workgroup” on page 217 for details about using workgroup folders.

  “Active Document” displays when **Always Display Active Document** has been selected from the Files list; the resources from the currently active file are always displayed.

To use a resource from a different file:

1. In the Resource Browser, do one of the following:
   - Select the desired open file or favorite from the **Files** list.
   - Select the desired library type (Vectorworks, User, or Workgroup) from the **Files** list. From the standard file open dialog box, locate the desired library file and click **Open**.

   The file name is displayed in the **Files** list, and its resources display in the resource display window.
2. Select the desired resource from the resource display window. Multiple resources can be selected and imported at the same time.
   - To use the resource immediately, double-click it to activate it, or select **Resources > Apply** or **Resources > Make Active**, if applicable. (Alternatively, drag the resource onto an object or to a location in the current file.) If symbol folders are present in the drawing, specify the location of the imported resource.
   - To import the resource for future use, select **Resources > Import**.
   - To reference the resource, select **Resources > Reference**. If the source file is not currently referenced by this file, specify the new reference information. (See “Referencing Resources” on page 213.)

The resource is added to the current file’s Resource Browser. (Default content is automatically imported into the current file at the point of use, and displays in the Resource Browser.)

When you access symbols or scripts within folders, the selected folder name is displayed in the Document Folder list, and its contents are displayed in the resource display window. To return to the main resource window, click on the Document Folder list, and then select **Top Level**. Alternatively, click the **Up One Level** button until the top level is reached.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Favorites Files
Finding Resources
Identifying Duplicate Resources
Working with Resources
Using the Resource Browser

Using Favorites Files

For quick access, the Resource Browser can store links to Vectorworks files that contain resources that you use frequently. You can then import resources from a **Favorites** file into any Vectorworks file and not have to search for the file again. Remove the link when the file is of no further use. Favorite files are remembered for future sessions.

Use the **Files** menu in the Resource Browser to create, use, and manage your **Favorites** list. Alternatively, add files (or aliases or shortcuts to the files) to the appropriate Favorites folder manually.

Making the Current File a Favorite

To make the current file a favorite:

In the Resource Browser, select **Files > Add Current to Favorites**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The file is added to the **Files** list under **Favorites**.

The file must be saved to be added as a favorite.

Making an Unopened File a Favorite

To make an unopened file a favorite:

1. In the Resource Browser, select **Files > Add New Favorite Files**.

   The standard Open dialog box opens.

2. Select the file to make a favorite, and then click **Open**.

   The file is added to the **Files** list under **Favorites**.

   The file must be saved in the same version of Vectorworks software as the current file to be added as a favorite.

Multiple files can be selected and added as favorites at the same time; alternatively, press Ctrl+A (Windows) or Command+A (Mac) to select all files at once.

Opening the Current Favorite

To open the current favorite:

1. In the Resource Browser, select the favorite file to open from the **Files** list.
2. Select **Files > Open Current Favorite**.

   The file opens.

Refreshing Favorites from Disk

To update the resources from all favorite files:

In the Resource Browser, select **Files > Refresh Favorites from Disk**.

All current favorite files are reloaded into the Resource Browser, to reflect any changes that occurred since they were loaded when this Vectorworks session began.

Revealing the Location of the Current Favorite

To reveal the location of a favorite file:

1. In the Resource Browser, select the favorite file to reveal from the **Files** list.

   The file name is displayed as the current file in the **Files** list, and its resources display in the resource window.

2. Select **Files > Reveal Current Favorite**.

   Either Windows Explorer or Mac Finder opens to the folder that contains the file.

Removing Favorites

To remove a single favorite file:

1. In the Resource Browser, select the favorite file to delete from the **Files** list.
2. Select **Files > Remove Current Favorite**.

   The file is removed from the **Favorites** list.

To remove all favorite files:

1. In the Resource Browser, select **Files > Remove All Favorites**.

   A confirmation dialog box opens.

2. Click **Yes** to remove all favorites.

   All favorite files are removed from the **Favorites** list.

Adding a Favorite Manually

To create a favorite manually (not through the Resource Browser):

1. Place the file that you want to appear on the **Favorites** list (or a shortcut or alias that points to the file) in one of the following folders.

Favorites Folder	Effect on Favorites Lists
[User]/Libraries/Favorites	The file appears only in your **Favorites** list ([User] is the user data folder specified in your Vectorworks preferences). See "User Folders Preferences" on page 57 for more information about user folders.
[Workgroup]/Libraries/Favorites (Vectorworks Design Series required)	The file appears in the **Favorites** list of any user who has the workgroup folder set up in Vectorworks preferences

   The file is added to the **Files** list under **Favorites**.

2. In the Resource Browser, select **Files > Refresh Favorites from Disk**.

   Only files that are in your user Favorites folder can be removed with the commands in the Resource Browser. (See "User Folders Preferences" on page 57 for more information.) Files that were placed in a workgroup Favorites folder must be removed manually.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Accessing Existing Resources
Finding Resources
Identifying Duplicate Resources
Working with Resources
Using the Resource Browser

Finding Resources

There are three ways to quickly access specific resources through the Resource Browser:

- Use the Resource Browser's incremental name search feature to locate and select a resource that is already in the document.
- Use the Resource Browser's **Find Resource** command to search for a specific resource name in the current file or in another file in a specified file path. When the resource is found, it can be selected from the Resource Browser.
- Find an instance of the resource in the drawing, and then Right-click (Windows) or Ctrl-click (Mac) and select **Locate in Resource Browser** from the context menu to select the resource in the Resource Browser.

Incremental Search

To quickly access a resource that is already in the document, click in the resource display window and begin to type the resource's name. The resource display window scrolls to the first resource or folder that begins with the letter(s) you entered. The resource type (such as Textures or Worksheets) must be shown in the browser window to be part of the search. If the resource is in a folder, select the folder from the File folder list first.

Finding a Resource

Use the **Find Resource** command to quickly locate a resource in any file created with the current version of the program, and optionally to select that resource in the Resource Browser.

To locate and use a resource:

1. In the Resource Browser, select **Resources > Find Resource**.

   The Find Resource dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Click to show/hide the parameters.

Parameter	Description
Find resource names containing	Enter the full or partial resource name to find
Search Location	
Current file	Select to search for the resource in the current file
Files on disk	Select to search for the resource in the displayed path. To search for the resource in a different folder, click **Choose**; select the folder to search in, and then click **Open** (Mac) or **OK** (Windows) to return to the Find Resource dialog box.
Include subfolders	Select whether to include subfolders in the search process
Options	
Resource types	Select to search for all resource types or for one specific resource type
Match case	Select whether the search is case sensitive
Find whole words only	Select this option to search for occurrences that are whole words only, and not part of a larger string of text

2. Enter the search criteria and click **Find**.

   The Find Resource Results dialog box displays the resource name, type, and file location for all resources that match the specified search criteria. Select the resource name and click **Select** (or double-click the resource name). If the selected resource is in a different file, the Resource Browser temporarily displays all the resources of that type (such as Hatches) that are contained in the file. The resource you selected is highlighted.

3. Once the resource is found, there are various ways to use it:

   - To use the resource immediately, double-click it to activate it, or in the Resource Browser select **Resources > Apply** or **Resources > Make Active**, if applicable. (Alternatively, drag the resource onto an object or to a location in the current file.) If symbol folders are present in the drawing, specify the location of the imported resource.
   - To import the resource for future use, select **Resources > Import**.
   - To reference the resource, select **Resources > Reference**. If the source file is not currently referenced by this file, specify the new reference information. (See “Referencing Resources” on page 213.)

The resource is added to the current file’s Resource Browser. (Default content is automatically imported into the current file at the point of use, and displays in the Resource Browser; see “Resource Libraries” on page 219.)

Locating Specific Resources

Resources used in a drawing can be quickly located and selected in the Resource Browser with the **Locate in Resource Browser** context menu command.

To locate a resource from the drawing:

1. Move the cursor over the resource (or object containing a resource, such as a rectangle with a hatch) to be located. Right-click (Windows) or Ctrl-click (Mac) and select **Locate in Resource Browser** from the context menu.

   The command name dynamically updates to reflect the resource; if the cursor is over an image, or an object with an image resource applied is selected, the command displays as **Locate Image in Resource Browser**. Textures applied to object parts can also be located (Renderworks required). Most resource types can be quickly located by this method, except for record formats, symbol folders, scripts, and script palettes.

2. The Resource Browser automatically navigates to, and selects, the specific resource.

Accessing Existing Resources
Working with Resources
Identifying Duplicate Resources
Using the Resource Browser

Identifying Duplicate Resources

When a resources name has been duplicated in the current file, the program checks to see if any difference exists between the two resources.

- If the two resources are identical, only one is listed for selection
- If there are variations between the resources containing duplicate names, the program appends the originating file name in parentheses to the default resource, and both items display for selection

Accessing Existing Resources
Finding Resources
Working with Resources
Using the Resource Browser
Working with Resources

D Exporting Custom Resources

If you create custom resources and reuse them frequently, you may want to organize them into resource libraries. You and other users can add the libraries to the Resource Browser favorites, or add them to the default content, and then easily use resources from them as needed. (For more information about resources, see “Using the Resource Browser” on page 221.)

In the Vectorworks Design Series products, use the export function of the Resource Browser to export one or more resources from the current document to another file, without opening the file. If an older version of the resource is already in the target document, it can be updated with the new version.

To export resources:

1. In the Resource Browser, select the resource(s) to export.
2. Select **Resources > Export**. Alternatively, right-click (Windows) or Ctrl-click (Mac) on the resource and select **Export Resource** from the context menu.

   The Export Resource dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Destination list	Displays Recent Export Files, Libraries, and Currently Open files. Either select a file from the window and click **OK**, or click **Browse** to select a file from another location. If the resource is to be exported one of the default content folders within the user folder (Libraries\Defaults), place the file within a subfolder if one exists.
Browse	Specifies a different location for the exported resource
Preserve folder hierarchy	During export, maintains the same folder structure the exported resource had in its source file

3. Click **OK**.

If the target file already contains a resource with the same name, the Resource Name Conflict dialog box opens with three options: replace the resource, rename the resource, or do not export that resource (any other selected resources will be exported, if there are no further name conflicts). Select an option, and then click **OK**.

If there is a name conflict between two resources of different types (such as a symbol and an image), the option to replace the resource in the target file is not available.

Symbols

Symbol Advantages

Objects can be saved as 2D (screen plane), 3D (2D planar or 3D), or hybrid (2D screen plane and 3D combined) symbol definitions. Vectorworks software also ships with thousands of symbols. Symbol definitions save the object properties, such as size, color, and class, within the symbol definition; these properties are retained each time the symbol is placed, and when a symbol is imported into a different drawing.

Symbols provide several advantages:

- **Smaller file sizes:** The symbol and its definition are stored only once in the drawing file. Placement information (location coordinates, rotation) is all that is required for each symbol instance.
- **One-time editing:** Changes to the symbol definition automatically update all the instances of the symbol in the drawing.
- **Attached database information:** The information associated with a symbol can be used to generate reports and worksheets. Information attached to a symbol is specific to that instance, allowing each instance to be edited individually.
- **Ease of import:** With the Resource Browser, importing symbols from one file to another is fast and easy, and any database information attached to the symbol is also imported.

  Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Symbol Types
Creating New Symbols

Symbol Types

In Vectorworks, you can create 2D and 3D objects. Symbols, which are converted objects, can also consist of 2D, 3D, or hybrid objects.

In addition, there are special symbol categories which indicate the symbol's behavior at placement. These categories are color-coded within the Resource Browser for identification. The category a symbol belongs to depends on the type of object converted to a symbol and the options selected at symbol creation.

Symbols can be nested within other symbols.

Symbol Type	Symbol Category
2D	2D symbols are composed exclusively of 2D screen plane objects. The instances of the symbol are considered planar, and may occupy the screen plane, layer plane, or a 3D plane.
3D	3D symbols are created from 2D planar objects and/or 3D objects, have a height (Z coordinate) as well as widths and lengths (X and Y coordinates). These objects display flat in Top/Plan view. However, they retain their 3D properties. Symbols created from 3D objects that are not hybrid objects appear flat in 2D views, but show dimension in 3D views.
Hybrid	A hybrid symbol contains both a 2D screen plane object and a 3D component, and displays correctly according to the view. The advantage of working with hybrid symbols is that 3D models can automatically be created from 2D drawings, or vice versa. For example, a hybrid door symbol displays as a "swing arc" in Top/Plan 2D view and as a fully formed door in a 3D view. Hybrid symbols must be inserted in the layer plane.
Black	This is the most common type of symbol, the static symbol. Its parameters are saved within the symbol definition, and set at placement. Changes made to the symbol definition affect all instances of the symbol.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Symbol Type	Symbol Category
Blue	When placed, this type of symbol is converted to a group. Any changes made to the symbol definition later have no effect on the group. At symbol creation, select **Convert to Group** in the insertion options to specify a "blue" symbol.
Red	When placed, this type of symbol is converted to a plug-in object. It has a specific insertion behavior (point, linear, rectangular, or path) and set parameters, and it can be modified, with many variations of the same object in the file. At symbol creation, select **Convert to Plug-in Object** in the insertion options to specify a "red" symbol. Changes to a "red" symbol definition affect future instances, but not existing ones.
Green	This is a page-based symbol. Typically these symbols are annotation symbols that are scaled relative to the page size, such as a North arrow indicator.

Plug-in Objects

Plug-in objects have all the power of standard symbols, with the added advantage of being customizable. Unlike symbols, plug-in objects have the option of being placed onto the drawing and remaining modifiable. This is useful if the drawing needs to contain many different variations of the same object.

Some tool sets and libraries contain plug-in objects; for example, the **Scale Bar** tool, located in the Dims/Notes tool set, inserts a plug-in object. In addition to the tool sets, pre-defined plug-in objects are available in the Libraries folder (in subfolders beginning with the word Object or Objects), and are accessed through the Resource Browser.

When a plug-in object from a tool set is inserted, an object properties dialog box may open the first time the item is placed in the drawing. The properties in this dialog box set the default values for the object during this session. Modify the properties prior to inserting the object, or accept the default values and click **OK**. Object instances can be modified through the Object Info palette after insertion.

Custom plug-in objects can be created through the **Tools > Plug-ins > Plug-in Manager**; see "Creating Scripted Plug-Ins" on page 1782.

In addition, a symbol can be saved as a "red" symbol that becomes a plug-in object upon insertion.

There are four different types of plug-in objects: point, linear, rectangular, and path. Each type is different in how it is placed in the drawing and edited.

> Linear and rectangular objects cannot be inserted directly into a wall. However, once placed in the drawing, they can be dragged onto a wall to insert them.

Point Plug-in Objects

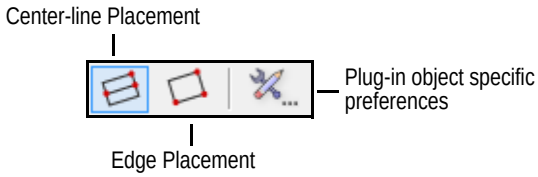
Point plug-in objects are placed by a single click in the drawing to specify the location, and then a second click to set the rotation angle. This is the same way symbols are placed using the symbol insertion tool. A preview image of the object is visible at the cursor location. Point plug-in objects cannot be edited (resized or rotated) with the cursor; they are edited using the Object Info palette.

Linear Plug-in Objects

Linear plug-in objects are placed with two clicks. The first click sets the beginning point and the second sets the endpoint of a line. The orientation of the object is determined by this line. The object can be resized or rotated by clicking on a reshape handle at either end of the line. It can also be edited using the Object Info palette.

Rectangular Plug-in Objects

Rectangular plug-in objects are placed by a sequence of three clicks in the drawing. There are two different placement modes which determine how these three clicks are interpreted.



- **Center-line Placement Mode:** The first click specifies the origin of the object, the second click specifies the length, and the third click defines half the width of the rectangular object. After the second click, the cursor displays feedback symmetrically on both sides of the center-line of the rectangle.
- **Edge Placement Mode:** The first click specifies one corner of the rectangular object, the second click determines the length, and the third click specifies the entire width.

When a rectangular plug-in object is selected, there are eight selection handles visible. The object can be resized by dragging these handles, or edited through the Object Info palette.

Path Plug-in Objects

Path plug-in objects are created with a sequence of several clicks that define the vertex points along a path. There are two different types of path plug-in objects based on the type of path that is used. A 2D path plug-in object uses a polyline path and a 3D path plug-in object uses a 3D NURBS curve for its path.

Path plug-in objects can be edited using the **Reshape** tool directly. They can also be edited through **Modify > Edit Group**, or the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating New Symbols
Inserting Symbols
Editing Symbols
Symbol Advantages
Record Formats
Global Symbol Commands

Creating New Symbols

The **Create Symbol** command creates symbols from 2D and/or 3D objects, including text. Symbols can also be created from other symbols, from plug-in objects, groups, and worksheets. At placement, the symbol can convert automatically into a group or plug-in object. Viewports cannot be made into symbols.

For more information on symbol types and color categories, see “Symbol Types” on page 237.

Item	Convert to	Result
2D screen object	2D symbol	2D black symbol, planar
3D object and/or 2D planar	3D symbol	3D black symbol, for use in a 3D view only
2D screen + 3D object	Hybrid symbol	Hybrid black symbol, for use in 2D plan and 3D models
Symbol	Symbol	Saves any changes to the current symbol attributes as a new, black symbol definition
	Group	Creates a new blue symbol definition, to be inserted as a group
	Plug-in object	Creates a new red symbol definition, to be inserted as a plug-in object
Group	Symbol	Creates a new black symbol definition, with multiple objects existing within a symbol container
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Item	Convert to	Result
Plug-in object	Symbol	Creates a new black symbol definition. This allows all instances to change by editing the plug-in object inside the symbol, even though the actual symbol instance cannot be edited.
	Plug-in object	Creates a new red symbol definition, to be inserted as a plug-in object. This allows an editable plug-in object to be placed in a drawing with saved parameters. For example, a door plug-in object, when saved with a width of 4’ and inserted as a plug-in object, is inserted with a width of 4’ rather than with the default width.
Worksheet	Group	Creates a new blue symbol definition, to be inserted as a group. For example, a worksheet can be saved as a preformatted report by saving it as a blue symbol definition.

To create a new symbol:

1. Select the object(s) to convert into a symbol.

   To create a hybrid symbol, select both the 2D and 3D objects, which become the 2D and 3D components of the symbol. In Top/Plan view, align the objects first (symbol alignment can be adjusted after creation with the **Edit Symbol** command). The symbol preview that displays in the Resource Browser is generated in the symbol view at creation or editing.

   If a hybrid symbol will be inserted into a wall, you can optionally specify the 3D wall hole component in addition to the 2D and 3D components; see “Adding a 3D Wall Hole Component to a Symbol Definition” on page 260.

   For 2D symbols inserted in walls, two loci can be specified as wall break locations. In Top/Plan view, place two loci at opposing locations with the object to convert, and select them all before converting. When the symbol is inserted into the wall, the wall breaks at the loci locations instead of the symbol bounding box.

2. Select **Modify > Create Symbol**.

   The Create Symbol dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Provide a name for the new symbol. Do not use single quotes in symbol names. Single quotes in names are reserved for use in scripts.
Insertion Point	Controls how the symbol is inserted
Plan Projection Center/ 3D Object Center/ Plug-in Object Origin	Sets the insertion point according to the object’s geometry. • If working in Top/Plan view, the insertion point is defined by the bounding box center. For 3D objects, the z coordinate is defined by the object's relation to the active layer plane. • If working in 3D view, the insertion point is defined as the center of the object's bounding cube. • If creating a symbol from a Plug-in object, the insertion point is defined to be the same as the object's insertion point.
Next Mouse Click	Sets the insertion point manually, with the cursor, after clicking **OK**
Units	Determines the symbol’s size units

Parameter	Description
Page-based	Sets the symbol size according to the page. When inserted, the symbol automatically scales relative to the page environment. This is useful for annotation objects that should always remain the same size on the "page," regardless of the layer scale.
World-based	Sets the symbol size according to constant world dimensions; its insertion size depends on the layer scale
Insert in Walls	Select whether the symbol will be insertable in walls. When Wall Insertion mode is enabled, objects that have the **Insert in Walls** option enabled will insert into walls (see "Inserting Symbols" on page 242).
On Centerline/On Edge	If **Insert in Walls** is selected, select how the symbol will be inserted. **On Centerline** snaps the symbol's insertion point to the center line of the wall. **On Edge** snaps the symbol's insertion point along either edge of the wall.
Wall Breaks	If **Insert in Walls** is selected, select how the wall will break around the symbol when it is inserted into a wall
Other Options	
Leave Instance In-Place	When selected, replaces the current selection with a symbol instance; when deselected, the object is removed from the drawing
Change 2D Objects from Layer Plane to Screen Plane	Converts any planar 2D objects in the layer plane into screen plane 2D objects, for the Top/Plan representation of the symbol. If deselected, any 2D planar objects in the layer plane that are part of the symbol will not display in Top/Plan view, so a hybrid symbol may not display properly in Top/Plan view.
Convert to Group	Converts the symbol into a grouped object when inserted, disassociating it from the original symbol definition. Converted grouped objects are identified in the Resource Browser with a blue name. Deselect to convert the symbol into a black, unmodifiable symbol; each instance is controlled by the symbol definition.
Convert to Plug-in Object	Converts the symbol into a plug-in object when inserted, allowing the current parameters to be pre-set when the plug-in object is inserted. Converted plug-in objects are identified in the Resource Browser with a red name. Deselect to convert the symbol into a black, unmodifiable symbol; each instance is controlled by the symbol definition.
Assign To Class	Select the class the symbol will be assigned to upon insertion. The symbol can be assigned to the active class or to another class present in the drawing; alternatively, create a new class by selecting **New.** When a new class is created, it does not automatically become the active class.

3. Click **OK**.
4. If the **Next Mouse Click** insertion point option was selected, click to specify the desired symbol insertion point.
5. Specify the folder location for the new symbol in the next Create Symbol dialog box.

   Click the folder to select it as the destination for the new symbol.

Click to show/hide the parameters.

Parameter	Description
Folder list	Displays the file's symbol folders; the selected folder becomes the destination for the new symbol
New Folder	Creates a new folder within the selected folder; specify the folder name and click **OK** to create the symbol folder
Don't show this dialog	Hides this dialog box if you only want to see it when a file already contains symbol folders

6. Click **OK**.

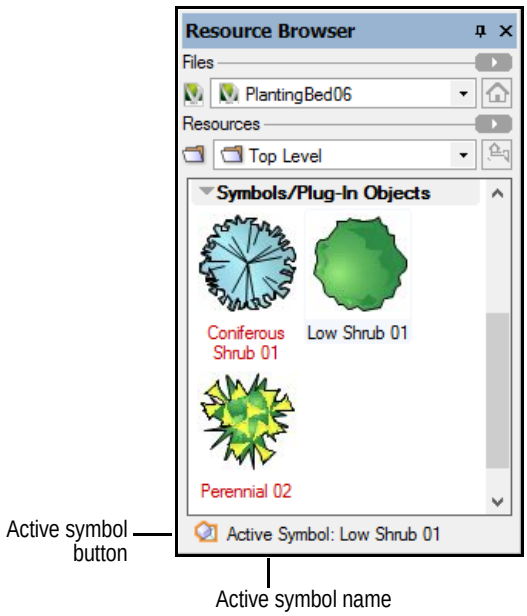
   The new symbol is added to the Resource Browser.

Inserting Symbols

Symbols are resources available from the Resource Browser. To open the Resource Browser, select **Windows > Palettes > Resource Browser**. Display the symbol to insert in the Resource Browser (see "Accessing Existing Resources" on page 229).

The currently active symbol is inserted; the active symbol's name displays at the bottom of the Resource Browser. Clicking on the active symbol button automatically displays the currently active symbol in the resource display window.

Symbols are inserted from the Resource Browser by dragging and dropping, or with the **Symbol Insertion** tool. Symbols can be inserted as individual objects or inserted to become part of a wall.



Drag and Drop Symbol Insertion Method
The Symbol Insertion Tool

Drag and Drop Symbol Insertion Method

To insert a symbol by dragging it from the Resource Browser:

1. Click the symbol in the Resource Browser and drag it to the desired location in the drawing. To insert the symbol into a wall, the symbol must have the **Insert in Walls** option enabled (see "Creating New Symbols" on page 239).

The symbol cannot be rotated during insertion, since the **Symbol Insertion** tool is not automatically activated. When a symbol is dropped onto a wall, round wall, or roof, Vectorworks inserts the symbol into the object using the default flip value and insertion point.

A symbol inserted by dragging and dropping is placed on the active layer plane by default. Press the Option key (Mac) or the Alt key (Windows) while dragging and dropping to place the symbol on the working plane.

2. The symbol is inserted.

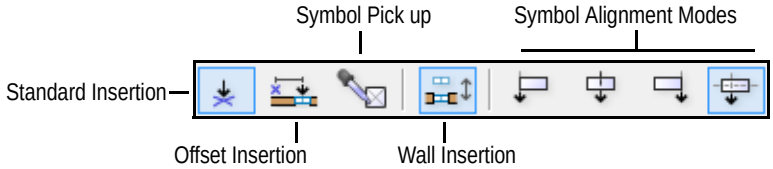
   If necessary, edit the symbol rotation or flip in the Object Info palette.

The Symbol Insertion Tool

Use the **Symbol Insertion** tool to place 2D, 3D, and hybrid symbols in the drawing. During insertion, a preview of the symbol is shown to aid in placement. Symbols can also be inserted by dragging them from the Resource Browser.

Each symbol has its own insertion point, a location on the symbol shown by crosshairs in the Edit Symbol window, that controls how the symbol is placed into the drawing. The insertion point is specified during the symbol's creation.

Hybrid symbols can only be inserted on a working plane that is parallel to the active layer plane. When inserting a hybrid symbol, if the working plane is not parallel to the active layer plane, the working plane is aligned with the active layer plane. (The working plane origin remains unchanged.)



Mode	Description
Standard Insertion	Inserts the symbol based on its specified insertion point, or aligned according to one of the symbol alignment modes
Offset Insertion	Inserts the symbol in a wall according to an offset reference point
Symbol Pick up	Designates a symbol from the drawing as the active symbol
Wall Insertion	Toggles between inserting a symbol or plug-in object into a wall with wall breaks, and inserting a symbol or plug-in object near or on a wall without breaking it. To prevent all symbol and plug-in objects from inserting into walls, turn off Wall Insertion mode. When Wall Insertion mode is enabled, objects that have the **Insert in Walls** option enabled will insert into walls.
Symbol Alignment Modes	These alignment modes apply to Standard Insertion mode, and temporarily override the insertion point. These modes change the alignment of the insertion point along the X axis of the bounding box surrounding the symbol. Alternatively, keep the original point as the insertion point.
Align Symbol Left	Moves the insertion point to the left edge of the symbol's bounding box, along the original X axis
Align Symbol Center	Moves the insertion point to the center of the symbol's bounding box, along the original X axis
Align Symbol Right	Moves the insertion point to the right edge of the symbol's bounding box, along the original X axis
Align Actual Insertion Point	Uses the symbol's originally specified insertion point

Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Standard Symbol Insertion Mode

 To insert a symbol:

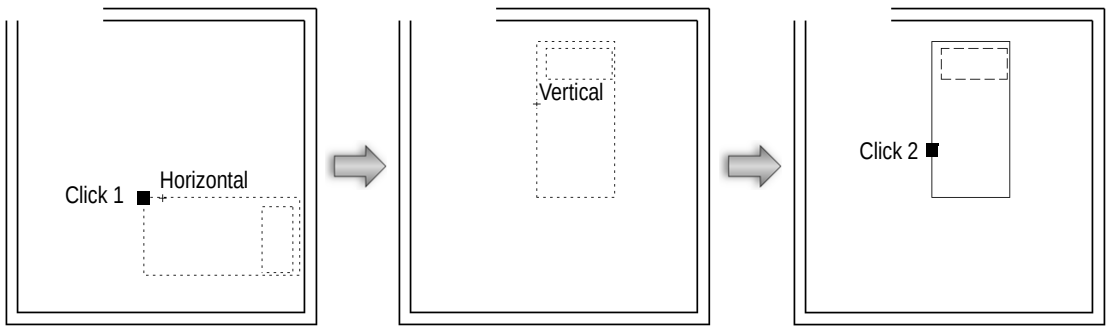
1. Select the desired symbol from the Resource Browser.
2. Select **Resources > Make Active**. The **Symbol Insertion** tool, on the Basic palette, automatically becomes active.

   Alternatively, double-click the symbol in the Resource Browser to make the symbol active and select the **Symbol Insertion** tool.
3. Click **Standard Insertion** from the Tool bar.

   To insert the symbol into a wall with breaks, enable **Wall Insertion** mode. The symbol also must have the **Insert in Walls** option enabled (see “Creating New Symbols” on page 239).
4. Click the desired alignment mode (see “Inserting Symbols” on page 242).
5. Click to set the location of the symbol.
6. If the symbol is not oriented properly, move the cursor slightly away from the insertion point; then move the cursor to rotate the symbol about its insertion point. In the Data bar, use the **A** (angle) and **WP A** (working plane angle) fields to help position the symbol.
7. Click a second time, without moving the mouse, to position the symbol exactly as inserted. Alternatively, move the cursor slightly away from the insertion point to rotate or flip the symbol about its insertion point.

   For symbols placed inside walls, moving the mouse flips the symbol about one of two axes: up and down, or left and right.

   In click-drag mode, click and hold the mouse to insert the symbol and still be able to rotate it. A quick click eliminates the ability to rotate or flip the symbol and locks the orientation of the symbol as it is placed.
8. Click to set the symbol.



To place another copy of the symbol in the drawing, move the cursor to another location and click. The selected symbol and **Symbol Insertion** tool remain active until another tool is selected.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Wall Insertion Mode

Offset Symbol Insertion Mode

Use Offset Insertion mode to insert a symbol into a wall at a specific distance from a reference point. The offset distance can be measured either to the symbol's insertion point or to a clicked point on the symbol.

 To insert a symbol into a wall with offset insertion:

1. Select the desired symbol from the Resource Browser. The symbol must have the **Insert in Walls** option enabled (see "Creating New Symbols" on page 239).
2. Select **Resources > Make Active**. The **Symbol Insertion** tool, on the Basic palette, becomes active.

   Alternatively, double-click the symbol in the Resource Browser to make the symbol active and to select the **Symbol Insertion** tool.
3. Click **Offset Insertion** mode.
4. Click the desired alignment mode.
5. Click to set the reference point from which the symbol is to be offset. The reference point does not have to be within the wall.

   The symbol preview displays when the cursor is over a wall.
6. Click to set the offset location of the symbol.
7. If the symbol is not oriented properly in the wall, move the cursor slightly away from the insertion point, and then flip the symbol about its axis to the correct position.

   In click-drag mode, click and hold the mouse to insert the symbol and still be able to flip it. A quick click eliminates the ability to flip the symbol and locks the orientation of the symbol as it is placed.
8. Click to set the symbol.

   The Enter Offset dialog box opens, displaying the distance between the clicked points. The fields available depend on whether the wall is straight or round.

Click to show/hide the parameters.

Parameter	Description
Offset	For straight walls, specify the offset distance. For round walls, specify either the **Distance** or the **Angle** between the reference point and the symbol. The distance is measured along the outer arc of the round wall.
Offset Using	Select whether to offset the symbol using the insertion point or the next mouse click on another point on the symbol

9. Click **OK**.
10. If the **Insertion Point** option was selected, the symbol is placed into the wall so that the insertion point is offset as specified from the reference point.

    If the **Next Click** option was selected, a witness line of the specified length or angle extends from the reference point to the end point. Click on the symbol to indicate the appropriate offset point and to place the symbol into the wall so that the clicked point is offset as specified from the reference point.

Symbol Pick Up Mode

The Symbol Pick up mode “picks up” any symbol already inserted into the drawing and makes it the active symbol. This avoids having to locate and select the symbol in the Resource Browser.

 To pick up and place a symbol:

1. Click the **Symbol Insertion** tool from the Basic palette.
2. Click **Symbol Pick up** mode.
3. Click a symbol in the drawing.

   The symbol becomes the active symbol. Note that the symbol clicked on is not highlighted (selection handles do not display).

   Either the Standard Insertion mode or Offset Insertion mode is automatically enabled, depending on the last mode used. Select a different insertion mode, if desired.
4. Insert the symbol according to the instructions for that mode.

To switch to Symbol Pick up mode quickly, press and hold the Option (Mac) or Alt (Windows) key while the **Symbol Insertion** tool is active, and then click on the desired symbol. The selected symbol is now ready to be inserted into the drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Standard Symbol Insertion Mode
Offset Symbol Insertion Mode

Wall Insertion Mode

When enabled, the Wall Insertion mode allows a symbol or plug-in object to be inserted into a wall. In addition, the symbol must have the **Insert in Walls** option enabled (see “Creating New Symbols” on page 239).

When Wall Insertion mode is disabled, a symbol or plug-in object is placed near or on top of a wall without being inserted directly into it.

A symbol or plug-in object inserted into a wall is automatically made parallel to the wall’s center line, by rotating it to match the wall’s angle. To remove a symbol from a wall, click and drag it out of the wall. Vectorworks automatically seals the cut in the wall.

For more information on inserting and moving symbols into, out of, and within walls, see “Bulb Flat” on page 458.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Drag and Drop Symbol Insertion Method
Inserting Symbols

Editing Symbols

Once a symbol instance is placed in a drawing, its information can be displayed in the Object Info palette. Select a symbol instance to display its properties. A symbol’s location can be adjusted, or its layer and class association can be changed by selecting a new class or layer from the appropriate list. A symbol can be replaced with another symbol, rotated in 2D or 3D space, and, for symbols located in walls, flipped, replaced, and repositioned.

Most of a black symbol’s physical attributes cannot be directly changed in the Object Info or Attributes palette. Instead, the components of a symbol must be accessed and edited through the Edit Symbol window. Changes made to a symbol definition affect all existing and future instances of that symbol, unless the symbol was inserted as a group (blue) or plug-in object (red).

If you delete a symbol definition and there are instances of that symbol in the drawing, you can specify whether to replace all instances with loci or to delete them completely.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Rotating Existing Symbols
Replacing Existing Symbols
Editing Symbol Definitions
Converting a Symbol Instance to a Group
Managing Symbols
Creating New Symbols
Inserting Symbols
Deleting Symbols
Inserting and Editing Symbols Within a Wall

Rotating Existing Symbols

Rotate 2D symbols in the screen plane by entering a rotation angle in the **Rot** field of the Object Info palette. Rotate 3D symbols by clicking the **Rotate 3D** button in the Object Info palette. This button accesses the Rotate Object in 3D dialog box for specification of the rotation angle, center, and axis; see “Rotate Tool” on page 1022.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbols

Replacing Existing Symbols

To exchange a symbol instance’s definition with another:

1. Select the existing symbol.
2. In the Object Info palette, click the **Replace** button.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a symbol, and select **Replace** from the context menu.

   The Choose a Symbol dialog box opens.
3. Select the desired new symbol.

   Only the symbols that are currently part of the active drawing are listed.
4. Click **OK**.

   The old symbol instance is replaced with the new symbol instance.

The new symbol uses the settings from the replaced symbol. Future occurrences of the new symbol are unaffected by these settings.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbols

Editing Symbol Definitions

Most of a black symbol’s physical attributes cannot be directly changed in the Object Info or Attributes palette. Instead, the components of a symbol must be accessed and edited through the Edit Symbol window. Changes made to a symbol definition affect all existing and future instances of that symbol, unless the symbol was inserted as a group (blue) or plug-in object (red).

Editing a symbol from the drawing area allows you to edit the symbol definition from within that symbol instance. Other objects in the drawing can be visible, grayed, or hidden depending on two settings in the Display tab of the Vectorworks preferences: **Show other objects while in editing modes** and **Gray other objects**. When other objects are visible, they are also snappable, which makes it is easier to edit the symbol quickly and accurately.

> The **Show other objects while in editing modes** preference does not work when editing a symbol definition from a flipped symbol instance; an alert message displays when this operation is attempted.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

When editing page-based (green) symbols from the drawing area, the scale of the symbol editing window is set to 1:1 so that the symbol definition is the same size as the symbol instance, for ease of editing. The active layer scale cannot be changed from the document context menu while in the symbol editing mode. This is an exception for page-based symbols only. Due to the scale adjustment, other objects that are not part of the symbol definition are not snappable unless they are also at a 1:1 scale in the active layer.

Referenced symbols can be edited or renamed in the target file, which changes the symbol definition in the source file. See “Referencing Resources” on page 213.

Editing behavior and results depend on the symbol type (see “Symbol Types” on page 237).

Symbol Type	Edit Behavior/Result
Black or green	Edit from the Resource Browser or from the drawing. Changes to the symbol, whether made to the definition or the instance, affect both existing and future symbol instances.
Blue	Edit from the Resource Browser. Changes to the definition affect future symbol instances only. A blue symbol inserted on the drawing as a group cannot be edited from the drawing in the Edit Symbol window (make edits directly from the Object Info or Attributes palette, and by editing the group). Changes to a drawing instance affect that instance only.
Red	Edit from the Resource Browser. Changes to the definition affect future symbol instances only. A red symbol inserted on the drawing as a plug-in object cannot be edited from the drawing in the Edit Symbol window (make edits directly from the Object Info or Attributes palette). Changes to a drawing instance affect that instance only.

To edit a symbol definition:

1. In the Resource Browser, select the symbol to edit, and select **Resources > Edit**.

   You can also right-click (Windows) or Ctrl-click (Mac) the symbol, and select **Edit 2D Component**, **Edit 3D Component**, **Edit 3D Wall Hole Component,** or **Edit Symbol Options.** For plant symbols, **Edit Definition** (Vectorworks Landmark required) is also available from the context menu.

   Alternatively, edit a black symbol instance from the drawing using one of the following methods:

   - Select the symbol and then select **Modify** > **Edit Symbol**.
   - Double-click the symbol.
   - Right-click (Windows) or Ctrl-click (Mac) the symbol, and select **Edit** from the context menu.
   - To edit the components of a hybrid symbol directly, right-click (Windows) or Ctrl-click (Mac) the symbol, and select **Edit 2D Component**, **Edit 3D Component**, or **Edit 3D Wall Hole Component** from the context menu.

2. Only one component of a hybrid symbol can be edited at one time. If the symbol is hybrid, the Edit Symbol dialog box opens. Insertion options and units can be specified when the symbol is edited from the Resource Browser, and additional double-click behavior for the dialog box can be specified when the symbol is edited from the drawing.

   Select the component to edit.

Click to show/hide the parameters.

Parameter	Description
2D/3D Component	Edits the 2D screen plane or the 2D planar/3D component of the symbol
3D Wall Hole Component	Edits the 3D wall hole component of the symbol definition (see “Adding a 3D Wall Hole Component to a Symbol Definition” on page 260)

Parameter	Description
Symbol Options	Edits the method of inserting the symbol into a wall, and whether symbols are world-based or page-based (see “Creating New Symbols” on page 239). To edit the symbol options of multiple symbols, see “Editing the Symbol Options of Multiple Symbols” on page 250)
Double-click	Sets the future behavior when double-clicking on a symbol in a drawing. Select whether to display the Edit Symbol dialog box, or directly edit the 2D or 3D component or insertion options. Select **Edits the Component based on current view** to automatically edit the 2D component if in Top/Plan view, or the 3D component if in one of the 3D views.
Use the edit view	Matches the Resource Browser thumbnail view to the selected editing view. For example, if a hybrid symbol’s thumbnail preview is set to Top/Plan, and the 3D component of the symbol is edited while in a Right Isometric view, the thumbnail view switches to Right Isometric to match.

3. Click **Edit**.
   - If **Symbol Options** was selected, the Symbol Options dialog box opens. See “Creating New Symbols” on page 239 for information on wall insertion options and units.
   - If one of the component options was selected, the Edit Symbol window opens, containing the symbol to be edited. A colored border around the drawing window indicates that you are in an editing mode. The **Exit Symbol** command becomes available from the **Modify** menu, and the **Exit Symbol** button is visible in the top right corner of the drawing window.

   To edit nested symbols, select **Modify > Edit Symbol** again.

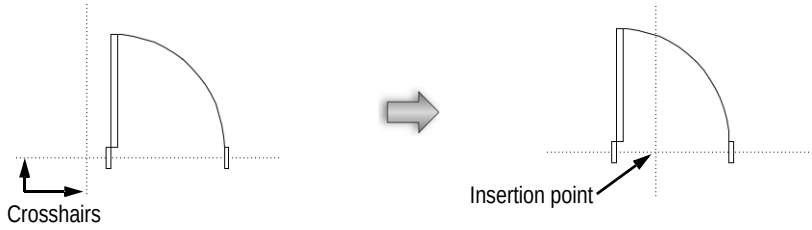
   The visibility of other objects when in editing mode is controlled by the **Show other objects while in editing modes option** in the Display tab of the Vectorworks preferences (see “Vectorworks Display Preferences” on page 50). To show the other objects in a less obtrusive way, also select the **Gray other objects** option. If the symbol was edited from the Resource Browser, other objects cannot be displayed; edit a symbol instance from the drawing to display other objects while in editing mode. See “Object Editing Mode” on page 1004 for more information). If a page-based (green) symbol is being edited, the editing mode scale has changed to 1:1, and while other objects that are not at a 1:1 scale may be visible at their own active layer scale, they are not snappable.

   The **Show other objects while in editing modes** preference does not work when editing a symbol definition from a flipped symbol instance; an alert message displays when this operation is attempted.

   If you are pasting 2D layer plane objects from the clipboard while editing a symbol, an alert allows you to assign those objects to the screen plane. Normally, select **Yes** so that the 2D components display correctly in Top/Plan view.

   In addition, if you are editing a 2D-only symbol and adding 3D objects (including 2D planar objects) or hybrid objects, or editing a 3D-only symbol and adding screen plane or hybrid objects, an alert informs you that you are creating a hybrid symbol. Portions of the symbol may not be visible in certain views. Similarly, if removing portions of a hybrid symbol during editing, you may be creating a 2D-only or 3D-only symbol which may not display as expected in certain views. Keep in mind that 2D objects that are part of a symbol must be in the screen plane to be visible in Top/Plan view; if they are planar, they will be visible in 3D views. 3D objects that are part of a symbol are not visible in Top/Plan view.

4. When editing components, make the symbol edits in the Attributes or Object Info palette. To edit the symbol insertion point, select all the components of the symbol, and relocate the components about the insertion point crosshairs. The intersection of the crosshairs gives the feedback segment Locus when encountered.



The other component of a hybrid symbol is not automatically adjusted to match changes made to the insertion point. It must be edited separately. Switch easily to the other component from the context menu.

5. After editing, click the **Exit Symbol** button (or select **Modify > Exit Symbol**) to update all instances of the symbol and return to the normal drawing mode.

   If the edited symbol is nested in other symbols, the **Exit Symbol** button returns back to the symbol container.

Editing the Symbol Options of Multiple Symbols

When multiple symbols are selected in the Resource Browser, it is possible to change the symbol options of all the symbols at one time.

To edit the symbol options of multiple symbols:

1. Select the symbols in the Resource Browser.
2. Select **Resources > Edit Symbol Options**.

   The Symbol Options dialog box opens.
3. The options are the same as those described in “Creating New Symbols” on page 239. However, the dialog box displays the current status of parameters for the selected symbols. When the parameter settings of the symbols are different, the option displays as blank, or a check box displays with an “indeterminate state;” any edits to that parameter affect all the selected symbols.

   If the selected symbols consist of a mixed set of symbol types, the available conversion options may differ (Convert to Group/Convert to Plug-in Object). Symbols are converted as described in “Creating New Symbols” on page 239.
4. Click **OK** to edit the symbol options.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbols
Editing Symbol Default Record Values
Object Editing Mode
Converting a Symbol Instance to a Group

Converting a Symbol Instance to a Group

Changes made to a black symbol definition affect all instances of that symbol in the drawing. The **Convert to Group** command changes a selected symbol into a group of Vectorworks objects, allowing edits to be made to that object without affecting the other instances of the symbol in the drawing. This command “disassociates” the symbol instance from its definition. The edited symbol can then be turned into a new symbol if desired.

Before using this command to edit a symbol instance that has been inserted into a wall, first drag the symbol outside of the wall. This avoids converting the wall as well.

To convert a symbol to a group:

1. In the drawing, select the symbol instance to edit.
2. Select **Modify > Convert > Convert to Group**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

If a symbol contains multiple levels of grouped objects, other symbols, or plug-in objects, the Convert to Group Options dialog box opens. Select the desired criteria for converting the symbol.

Click to show/hide the parameters.

Parameter	Description
Don't convert sub-objects to groups	Excludes any subobjects, such as nested symbols, from the conversion process
Convert nested symbols and plug-in objects	Converts nested symbols and/or plug-in objects within the symbol to individual objects within the group
Convert all sub-objects	Converts all objects within the symbol to individual objects within the group

Use caution when converting hybrid symbols. If in Top/Plan view, the 3D component of the symbol may be lost during the conversion. Similarly, in a 3D view, the 2D screen plane portion of the symbol may be lost.

3. Click **OK**.

   The symbol is converted into a group. To make changes to grouped items, select **Modify > Edit Group** or **Ungroup**.

   The object can be left as a grouped object or saved as a new symbol. Any changes to the original symbol definition do not affect this instance.

   Instead of repeatedly converting a symbol to a group, create a "blue" symbol which automatically converts to a group upon insertion.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbols
Creating New Symbols

Managing Symbols

Creating a New Symbol Folder

Create symbol folders to organize symbols within the Resource Browser.

To create a new symbol folder:

1. Select **Resources > New Resource** to display the list of new resource types.
2. Select **Symbol Folder**.

   The Assign Name dialog box opens.
3. Enter the name to assign to the new symbol folder.
4. Click **OK**.

   The new folder is added to the Resource Browser.

Moving a Symbol into a Folder

Group symbols according to topic or placement by moving them into folders within the Resource Browser.

To move a symbol into a folder:

1. Select the symbol in the Resource Browser.
2. Select **Resources > Move**.

   The Move Symbol dialog box opens. Click the folder to select it as the destination for the new symbol.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Click to show/hide the parameters.

Parameter	Description
Folder list	Displays the file's symbol folders; the selected folder becomes the destination for the symbol
New Folder	Creates a new folder within the selected folder; specify the folder name and click **OK** to create the symbol folder

3. Click **OK**.

   Vectorworks moves the symbol to the new folder.

Importing a Symbol or Symbol Folder

To import a symbol or symbol folder from another Vectorworks file:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.

2. Locate the file that contains the symbol or symbol folder to be imported.

   For information using the Resource Browser to locate resources in other Vectorworks files, see "Accessing Existing Resources" on page 229.

3. Select the symbol or symbol folder to be imported, and then select **Resources > Import**. (Alternatively, drag the symbol or symbol folder resource into the desired Vectorworks file.)

   The Import Symbol(s) dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Preserve folder hierarchy (symbol folder import only)	Imports the symbol folder, and any subfolders and symbols it contains, and maintains the same folder structure it had in its source file
Select destination folder	For symbol folders, imports the symbol folder, and any subfolders and symbols it contains, to the selected folder; for symbols, imports the symbol to the selected folder
Folder list	Displays the file's symbol folders; the selected folder becomes the destination for the symbol or symbol folder
New Folder	Creates a new folder within the selected folder; specify the folder name and click **OK** to create the symbol folder
Don't show this dialog when the current file has no symbol folders	If the current file has no symbol folders, select this option to hide this dialog box when you import symbols or symbol folders by dragging them into drawings in the future

4. Set parameters and click **OK**.

   If the target file already contains a resource with the same name, the Resource Name Conflict dialog box opens with three options: replace the resource, rename the resource, or do not import the resource. Select an option, and then click **OK**.

5. Click **OK**.

   The symbol folder and any symbols it contains are imported into the active Vectorworks file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a New Symbol Folder
Referencing Resources

Deleting Symbols

When a symbol resource is deleted, all instances of the symbol in the current document can be either deleted or replaced with loci to preserve the symbol locations.

To delete a symbol resource:

1. In the Resource Browser, select the symbol definition to delete, and select **Resources > Delete**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a symbol definition, and select **Delete** from the context menu, or select the symbol definition and press the Delete key.

2. An alert displays to confirm that the resource should be deleted. If there are any instances of the symbol currently in the document, specify whether to replace all instances with loci or to delete them completely. Click **Yes**.

   The symbol resource is deleted and each symbol instance is either deleted or replaced with a locus.

Inserting and Editing Symbols Within a Wall

Once walls have been created in a drawing, symbols and plug-in objects such as windows and doors can be added to them. After they are in the wall, the symbols may be moved or duplicated. When a 3D wall hole component has been added to a symbol, it can cut a hole of the desired shape in a wall.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

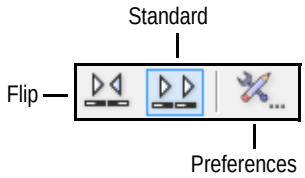
Inserting Symbols with the Duplicate Symbol in Wall Tool
Inserting Symbols in a Duplicate Array
Dragging and Dropping Symbols into Walls
Moving Symbols in Walls with the Selection Tool
Nudging Symbols in Walls
Moving Symbols in Walls with the Move Command
Moving Symbols in Walls with the Move by Points Tool
Editing Symbols in Walls
Adding a 3D Wall Hole Component to a Symbol Definition
Creating New Symbols
Editing Symbols
Inserting Symbols

Inserting Symbols with the Duplicate Symbol in Wall Tool

Use the **Duplicate Symbol in Wall** tool to easily place multiple copies of a symbol (or plug-in object) in a wall, such as doors or windows in an office building or electrical outlets along a wall.

> Symbols can also be duplicated within walls with the **Duplicate Array** command (see “Inserting Symbols in a Duplicate Array” on page 255) and with the **Move by Points** tool (see “Moving Symbols in Walls with the Move by Points Tool” on page 258).

The **Duplicate Symbol in Wall** tool has two modes.



Mode	Description
Flip	Flips the symbol during placement (for example, changes the direction that a door opens)
Standard	Places the symbol using the orientation at creation
Preferences	Specifies symbol placement information

 To place duplicate symbols in a wall:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.

2. Click the desired symbol from the list.

   The symbol must have the **Insert in Walls** option enabled (see "Creating New Symbols" on page 239).

3. Select **Resources > Make Active**.

   Activate a symbol already placed in the drawing with the **Symbol Pick up** mode of the **Symbol Insertion** tool (see "Symbol Pick Up Mode" on page 246).

4. Click the **Duplicate Symbol in Wall** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set

5. Select the placement mode from the Tool bar.

   This sets the symbol orientation. To place the symbol using the orientation in which is was created, click **Standard** mode. To flip the symbol when it is placed, click **Flip** mode.

6. Click **Preferences** from the Tool bar.

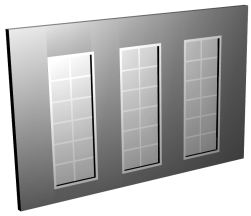
   The Duplicate Symbol In Wall dialog box opens. Indicate how to place the symbols.

Click to show/hide the parameters.

Parameter	Description
Start Offset	To set the first symbol position with the mouse, select **Next Click**; to set the first symbol position by a specified distance from a wall end, select the button next to the text box and enter a distance from the wall end
Spacing	To place successive symbols by mouse click, select **Next Click**; to place successive symbols a specified distance apart, select the button next to the text box and enter a distance between symbols
Copies	If selected, enter the specific number of symbol copies to insert into the wall

7. Click **OK**.
8. Click on the wall to indicate where the first symbol should be inserted. A preview of the symbols are shown along the wall to aid in placement. If inserting the first symbol by an offset value, click anywhere on the wall and the first symbol preview displays at the proper offset distance.
9. If spacing symbols by the next mouse click, move the mouse along the wall to set the spacing between symbols, and click again to insert the duplicate symbols. If spacing symbols by a specified distance, click again to insert the duplicate symbols. If a number of copies was specified, that number of symbols is automatically inserted.

   Move the mouse to the opposite side of the wall to flip the symbol direction.



Moving Symbols in Walls with the Selection Tool
Nudging Symbols in Walls
Moving Symbols in Walls with the Move Command
Moving Symbols in Walls with the Move by Points Tool
Editing Symbols in Walls

Inserting Symbols in a Duplicate Array

Use the **Duplicate Array** command to place multiple copies of a selected symbol into straight or round walls.

> Symbols can also be duplicated within walls with the **Duplicate Symbol in Wall** tool (see "Inserting Symbols with the Duplicate Symbol in Wall Tool" on page 253) and with the **Move by Points** tool (see "Moving Symbols in Walls with the Move by Points Tool" on page 258).

To create a duplicate array of symbols in a wall:

1. Select the symbol in the wall that is to be duplicated. The duplicates will be placed along the wall following the wall's blue direction arrow.
2. Select **Edit > Duplicate Array**.

   The Duplicate Array dialog box opens. The fields available depend on whether the wall that contains the selected symbol is straight or round.

Click to show/hide the parameters.

Parameter	Description
Direction	The preview window shows the direction the duplicate symbols will be placed in the wall. To reverse the direction of the duplication, click the button to the left of the window.
Number of Duplicates	Specify the number of copies of the original object to create
Offset Between Duplicates	Specify how far apart to place the duplicate symbols. For straight walls, enter the distance from the center of each symbol to the center of the next symbol in the wall. For round walls, specify either the distance or the angle between the duplicates. **Distance** is measured along the outer arc of the round wall. **Angle** is the angle between the center of each symbol to the center of the next symbol in the wall.
Z Offset Between Duplicates	Optionally, specify a difference in height on the wall between each symbol

3. Click **OK**, and the array of duplicate symbols is placed into the wall.

Moving Symbols in Walls with the Selection Tool
Nudging Symbols in Walls
Moving Symbols in Walls with the Move Command
Moving Symbols in Walls with the Move by Points Tool
Editing Symbols in Walls

Dragging and Dropping Symbols into Walls

To insert a 2D, 3D, or hybrid symbol in a wall by dragging it from the Resource Browser:

1. Click the symbol in the Resource Browser and drag it to the desired location in the wall.

   The symbol must have the **Insert in Walls** option enabled (see "Creating New Symbols" on page 239).

   The symbol cannot be rotated during insertion, since the **Symbol Insertion** tool is not automatically activated. When you drop a symbol onto a wall or round wall, the symbol is inserted using the default flip value.

2. If necessary, edit the symbol parameters in the Object Info palette.

Moving Symbols in Walls with the Selection Tool
Nudging Symbols in Walls
Moving Symbols in Walls with the Move Command
Moving Symbols in Walls with the Move by Points Tool
Editing Symbols in Walls

Moving Symbols in Walls with the Selection Tool

The **Selection** tool moves symbols (or plug-in objects) within walls. Several symbols can be selected and moved at once. When multiple symbols are selected, their common parameters can be changed in the Object Info palette.

Symbols within walls and symbols outside of walls cannot be selected at the same time.

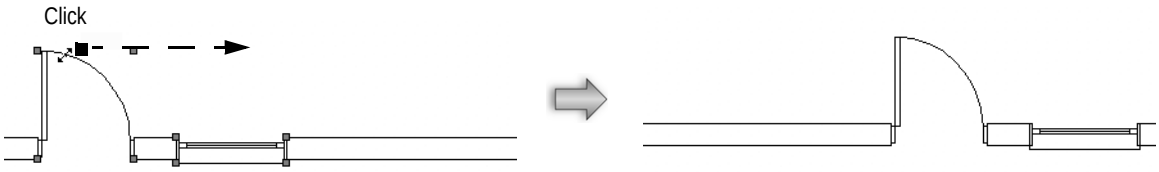
To move one or more symbols within a wall:

1. Click the **Selection** tool from the Basic palette.

   Toggle Wall Insertion mode on or off, depending on the desired result.

Action	Description
Select one or more symbols inserted in a wall, with **Wall Insertion** mode on	Inserted symbols can be dragged out of the wall (and into another wall if desired)
Select one or more symbols not inserted in a wall, with **Wall Insertion** mode on	Selected symbols can be inserted into a wall by dragging them over the wall
Select one or more symbols inserted in a wall, with **Wall Insertion** mode off	The symbols are moved within the wall. When one of the objects in the selection reaches the end of the wall, none of the objects can be moved farther in that direction.

2. Select the symbol or symbols.
3. Position the cursor near the symbol to be moved. When the resize cursor displays, drag the symbol(s).



Nudging Symbols in Walls
Moving Symbols in Walls with the Move Command
Moving Symbols in Walls with the Move by Points Tool
Editing Symbols in Walls

Nudging Symbols in Walls

Objects within a single wall can be nudged a small distance with a keyboard shortcut defined in the Edit tab of the Vectorworks Preferences (see “Edit Preferences” on page 49).

To nudge one or more symbols in a wall:

1. Select one or more symbols with the **Selection** tool; if multiple symbols are selected, they must be in the same wall.
2. Use the keyboard shortcut for nudging objects to move the symbols as needed. When one of the objects in the selection reaches the end of the wall, none of the objects can be moved farther in that direction.

Moving Symbols in Walls with the Selection Tool
Moving Symbols in Walls with the Move Command
Moving Symbols in Walls with the Move by Points Tool
Editing Symbols in Walls

Moving Symbols in Walls with the Move Command

Symbols within a single wall can be moved with the **Move** command.

To move one or more symbols in a wall:

1. Select the symbol(s) to move with either the **Selection** tool; if multiple symbols are selected, they must be in the same wall.
2. Select **Modify > Move > Move**.

   The Move Selection dialog box opens. The fields available depend on whether the wall that contains the selected symbol(s) is straight or round.

Click to show/hide the parameters.

Parameter	Description
Direction	The preview window shows the direction the symbol(s) will be moved in the wall. To reverse the direction of the move, click the button to the left of the window.
Offset	For straight walls, specify the distance along the wall to move the object(s). For round walls, specify either the **Distance** or the **Angle** between the old and new locations. The distance is measured along the outer arc of the round wall.

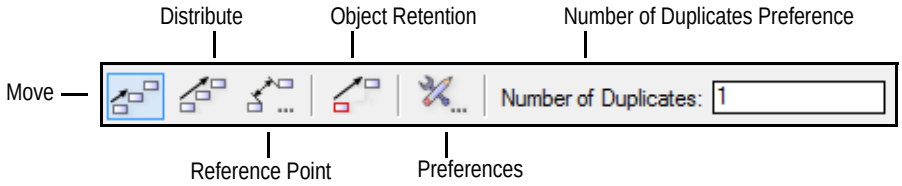
3. Click **OK**, and the symbols are moved as specified.

Moving Symbols in Walls with the Move by Points Tool

Symbols within a single wall can be moved, duplicated, and distributed along a specified distance with the **Move by Points** tool. The tool works in both 2D and 3D views.

> Symbols can also be duplicated within walls with the **Duplicate Symbol in Wall** tool (see "Inserting Symbols with the Duplicate Symbol in Wall Tool" on page 253) and with the **Duplicate Array** command (see "Inserting Symbols in a Duplicate Array" on page 255).

The **Move by Points** tool has the following modes.



Mode	Description
Move	Moves and duplicates selected symbols according to the distance and direction specified by two mouse clicks
Distribute	Moves and distributes duplicate symbols between the points specified by two mouse clicks
Reference Point	Moves selected symbols according to a clicked reference point and a specified offset distance from that point; the Preferences settings are not applicable when this mode is selected
Object Retention	Keeps the original symbols; this is the same as selecting the **Retain** option in the tool preferences

Moving Symbols in Walls with the Move or Distribute Mode

The Move and Distribute modes work the same way for symbols in walls as they do for other objects, except the vector that is specified by clicking is assumed to be along the wall. (See "Moving Objects with the Move or Distribute Mode" on page 1007 for details.)

To move, duplicate, and distribute selected symbols in a wall:

1. Select the symbol(s) in a wall to move and/or duplicate; if multiple symbols are selected, they must be in the same wall.
2. Click the **Move by Points** tool from the Basic palette and click **Preferences** from the Tool bar.

   The Move by Points Settings dialog box opens. Select the desired settings and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Number of Duplicates	Specifies the number of copies of the original object(s) to create (must be at least one); same as the **Number of Duplicates** field on the Tool bar

Parameter	Description
Original Object	
Retain	Keeps the original symbol(s); this option performs the same function as selecting the **Object Retention** button on the Tool bar
Leave Selected	Keeps the original symbol(s) selected for further action

To select additional symbols or to change the currently selected symbol to move, duplicate, or distribute, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking additional objects or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd is being pressed.

3. Select either the Move or Distribute mode, depending on the desired outcome.
4. Click once to indicate the vector start point, and click again to indicate the vector end point.

   Symbols are duplicated or moved within the wall, in the direction and distance specified. If the wall is not long enough to accommodate the specified duplicates and spacing, the number of symbols and the distance between them is adjusted automatically.

Moving Symbols in Walls with the Reference Point Mode

Reference Point mode can work two ways; the first click can either indicate the object to be moved or the reference point from which the object will be offset.

To move selected symbols in a wall using a reference point:

1. Select the symbol(s) to move; if multiple symbols are selected, they must be in the same wall.
2. Click the **Move by Points** tool from the Basic palette, and select the Reference Point mode from the Tool bar.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the symbol and select **Set Position** from the context menu. The **Move by Points** tool is automatically selected, in Reference Point mode.

   To select additional symbols or to change the currently selected symbol to move, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking additional objects or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd is being pressed.
3. Click to indicate either the reference point, or a point on the selected symbol.

   The Enter Offset dialog box opens, displaying the distance between the clicked points. The fields available depend on whether the wall is straight or round.
4. For round walls, select whether to offset the symbol by **Distance** or by the **Angle** along the arc of the wall.
5. Change the offset distance or angle.
6. Select whether the first click made was the reference point or a point on the symbol to be moved.
7. Click **OK**. The selected symbol is moved along the wall as specified.
8. The setting for how the first click is interpreted remains set until it is changed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Symbols in Walls with the Selection Tool
Nudging Symbols in Walls
Moving Symbols in Walls with the Move by Points Tool
Editing Symbols in Walls
Moving Objects by Clicking with the Move by Points Tool
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbols in Walls

Once a symbol has been placed in a wall, you can flip the symbol in the wall, reposition it, or completely replace it. Symbols can also be dragged completely out of the wall.

To change how a symbol interacts with a wall:

1. Select **Window > Palettes > Object Info**.

   The Object Info palette opens.

2. Select the desired symbol or symbols with the **Selection** tool.

   The Object Info palette displays the symbol's information. If several symbols are selected, only the common information can be changed.

3. Change the symbol's settings as necessary.

Click to show/hide the parameters.

Parameter	Description
Insert	Changes the insertion point location in relation to the symbol position
Break	Changes the wall break style where the symbol is inserted
Height	Changes the height of the symbol in the wall
Flip	Click to flip through a series of four rotations until the desired orientation is reached.  Alternatively, Right-click (Windows) or Ctrl-click (Mac) on the symbol and select **Flip** from the context menu.
Set Position	Activates the **Move by Points** tool in Reference Point mode; see "Moving Symbols in Walls with the Reference Point Mode" on page 259
Replace	Opens the Choose a Symbol dialog box to allow selection of a replacement symbol; see "Replacing Existing Symbols" on page 247

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbols
Adding a 3D Wall Hole Component to a Symbol Definition

Adding a 3D Wall Hole Component to a Symbol Definition

By adding a 3D wall hole component to a symbol definition, the symbol has the ability to cut holes of any shape in a wall. The geometry drawn to define the shape of the 3D hole can consist of any solid shape or shapes. If a wall hole component is included in a symbol definition, any existing 3D loci in the 3D component of the symbol are ignored.

To add a 3D wall hole component to a symbol definition:

1. Select a black symbol instance from the drawing. Right-click (Windows) or Ctrl-click (Mac) on the symbol, and select **Edit 3D Wall Hole Component** from the context menu to enter wall hole component editing mode.

   It is also possible to select the symbol to edit from the Resource Browser, and select **Resources > Edit**, and then select **3D Wall Hole Component** from the Edit Symbol dialog box. However, editing the symbol instance from the drawing allows you to see the drawing context when adding the wall hole component.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

A colored border around the drawing window indicates that you are in an editing mode. The **Exit Wall Hole Component** command becomes available from the **Modify** menu, and the **Exit Wall Hole Component** button is visible in the top right corner of the drawing window.

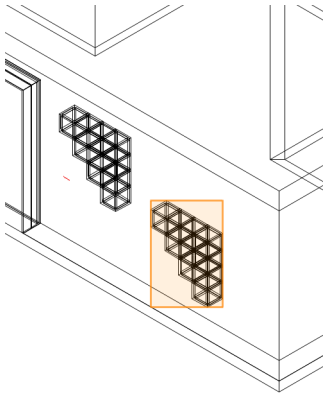
2. Any solid shape(s) can be drawn to define the hole shape. The geometry must intersect the wall.

   Select the **2D Polygon** tool or **Polyline** tool from the Basic palette and trace the symbol outline to define the hole shape. Select **Model > Extrude** to create a 3D wall hole shape. Alternatively, create 3D wall hole geometry directly using 3D modeling tools. The wall hole geometry has a red pen style; this can be changed from the Attributes palette if desired.

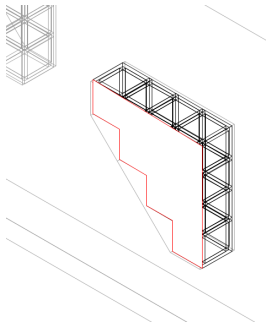
   Other drawing objects display, and can be snapped to, while in the editing mode (the Vectorworks display preference **Show other objects while in editing modes** must be enabled). The symbol being edited is displayed with its pen style attributes, to distinguish it from the rest of drawing.

3. Click **Exit Wall Hole Component** to return to the design layer.

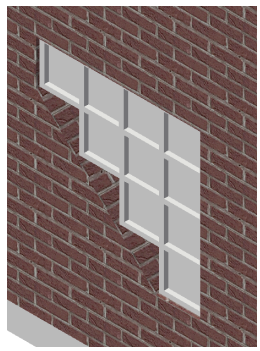
   Since the wall hole component was added to the symbol definition, all instances of the symbol will cut holes in walls according to the geometry drawn.



Right-click (Windows) or Ctrl-click (Mac) and select Edit 3D Wall Hole Component from the context menu to enter wall hole component editing mode



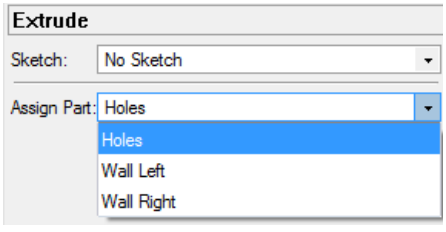
In the editing mode, draw the geometry to create the 3D wall hole, and extrude it to create the wall hole shape

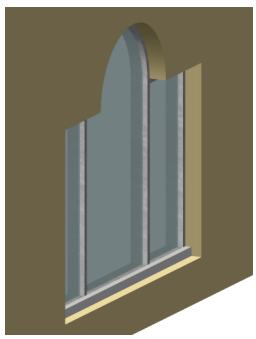


The 3D geometry creates the hole of the desired shape in the wall

Applying Renderworks Textures to Wall Hole Components

Special texture parts for wall hole components can be applied from the Object Info palette (Renderworks product required). See “Applying a Texture to an Object” on page 1530. While editing the wall hole component, the **Assign Part** list on the Render tab of the Object Info palette lists three options for the different textures that can be applied to wall hole faces: Holes, Wall Left, and Wall Right.





Texture Part	Result
Holes	Applies the texture specified for the wall's "Holes" part to the wall faces created by the cutting object
Wall Left	Applies the texture specified for the wall's "Left" part to the wall faces created by the cutting object
Wall Right	Applies the texture specified for the wall's "Right" part to the wall faces created by the cutting object

Each wall hole component can only have one texture part defined. Therefore, up to three different cutting objects would be necessary to apply three different texture parts to the wall hole faces.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying a Texture to an Object
Editing Symbol Definitions

Record Formats

Record formats, which store a wide range of data (like price or part numbers), can be attached to any object or symbol. Records attached to an object or symbol definition become a permanent part of it, remaining with the object or symbol even when it is imported or cut and pasted into another drawing. Several record formats can be attached to a single object or symbol, and record values can be individually changed for each object to which the record is attached. Record formats can be split and merged, to manage unwieldy or unnecessary formats.

In the Vectorworks Design Series products, record formats can be linked to an external database for automated, two-way communication; see "Database Connectivity" on page 1795.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Record Formats
Attaching Record Formats to Symbols and Objects
Editing Record Formats
Splitting and Merging Record Formats
Linking Text to Record Formats
Modifying Objects by Record Value

Creating Record Formats

Creating record formats in a drawing file is an important step to creating meaningful database rows in worksheets (see "Entering Data in Database Rows" on page 1339).

To create a new record format:

1. In the Resource Browser, select **Resources > New Resource**.
2. Select **Record Format**.

   The Create Record Format dialog box opens.
3. Enter the **Name** of the Record Format.
4. Click **New**.

   The Edit Field dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Enter a name for the field, up to 63 characters

Parameter	Description
Type	Select the type of field
Integer	Select to use whole numbers ranging from -32,768 to 32,767. Using **Integer** requires less memory than **Number**.
Boolean	Select to use a data value of either True or False
Text	Select to enter a string of characters (default option), such as a word or a sentence
Number	Select to use numbers outside the range of **Integer**, fractions or decimals, or to specify a number format. Click **Format** to define the number format in the Number Format dialog box; click **OK** to return to the Edit Field dialog box.
General	The default format
Decimal	Uses decimal numbers; enter a value for the number of decimal places, and if desired, select to use commas as separators
Scientific	Uses scientific numbers; enter a value for the number of decimal places
Fractional	Uses fractional numbers; enter the rounding value for fractions
Percentage	Uses percentages; enter a value for the number of decimal places
Dimension	Uses dimension numbers
Dimension Area	Uses dimension area format and displays the specified area units after the number
Dimension Volume	Uses dimension volume format and displays the specified volume units after the number
Angle	Determines the accuracy of angles and measurement system applied; measurement system is degrees, minutes, and seconds or decimal numbers up to eight decimal places
Date	Uses dates; select the desired date format from the list
Default	Enter the data value in the **Default** text box, if desired

5. Set the parameters and click **OK** to return to the Create Record Format dialog box.
6. For each additional field to add, repeat steps 4 through 6.
7. Click **OK** to return to the drawing area.

   When an object is selected, the Data tab of the Object Info palette displays all records contained in the current drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Attaching Record Formats to Symbols and Objects
Editing Record Formats
Splitting and Merging Record Formats
Linking Text to Record Formats
Modifying Objects by Record Value
Formatting Worksheet Cells
Database Connectivity

Attaching Record Formats to Symbols and Objects

Once a record format has been created, it can be attached to any object or symbol in the same drawing file as the record format. The Data tab of the Object Info palette indicates all record formats currently available to attach.

There are several methods of attaching record formats to symbols. Attach a record format to a single symbol instance, or to an object in the drawing, without affecting previous or future instances. Attach a record format to a symbol
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

definition that then applies to each symbol added to the drawing afterwards, or attach a record to a symbol definition and affect all existing and future symbol instances.

For global symbol changes, use the utility described in “Attaching Records” on page 273.

Attaching Record Formats to a Single Symbol Instance or Object

To attach (or detach) record formats to a single symbol or object in the drawing using the Object Info palette:

1. Select the symbol.
2. Select **Window > Palettes > Object Info**.

   The Object Info palette opens.
3. Select the Data tab.

   The Object Info palette lists all record formats in the drawing.
4. In the check box next to the desired record format(s), click to attach or deselect to detach the record format.

   If attaching a record format, an X displays in the box and the record is attached to that instance of the symbol or object.

   If detaching a record format, confirm the procedure.

To attach record formats to a symbol or object in the drawing using the Resource Browser:

1. Select the symbol(s) in the drawing.
2. From the Resource Browser, select the record format to be applied. Right-click (Windows) or Ctrl-click (Mac) and select **Apply** from the context menu. (Alternatively, double-click the record format resource to apply it to the selection or drag the record format resource onto a symbol or object.)

Attaching Record Formats to a Symbol Definition

There are two methods of attaching records to symbol definitions. The first method edits the symbol definition and allows records to be attached as well as default record values to be specified for the symbol definition. Existing symbol instances in the drawing are not affected. The second method attaches records to symbol definitions through the Resource Browser; both existing and future symbol instances are affected.

To attach record formats to a symbol definition by editing the symbol definition:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.
2. Select a symbol definition.
3. Select **Resources > Edit**.
4. The Edit Symbol dialog box opens. Select the symbol component to edit, and click **Edit**.
5. In the Edit Symbol window, deselect all by clicking in an empty area of the window.
6. In the Object Info palette, click on the Data tab.

   When no items are selected, the Data tab displays *SYMBOL DEFAULTS* at the top.
7. Select the record to attach.

   To attach multiple records to the same symbol definition, click on each of the records.

   The default Record Info value(s) of each Record Field can be set from the Data tab, affecting this symbol definition only without changing the record format resource; see “Editing Symbol Default Record Values” on page 267.
8. Click **Exit Symbol** at the top right of the Edit Symbol window.

The attached record(s) is included with the symbol each time the symbol is placed in the drawing or imported into another drawing. Symbols already present in the drawing are unaffected.

To attach record formats to a symbol definition from the Resource Browser:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.

2. Select a symbol definition.
3. Select **Resources > Attach Record**.

   The Attach Record dialog box opens.

4. The record formats present in the drawing are listed. Select the records to attach to the symbol definition, and click **OK**.
5. Existing symbol instances, as any future symbols placed from the symbol definition, have the record format attached.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Record Formats
Editing Record Formats
Splitting and Merging Record Formats
Linking Text to Record Formats
Modifying Objects by Record Value
Database Connectivity
Global Symbol Commands
Working with Resources

Editing Record Formats

Record formats can be edited in a variety of ways, depending on the desired outcome.

Method	Result
Edit the record format resource, changing the field list and/or default field values (see “Editing Default Record Formats” on page 266)	Future objects or symbols with the record attached reflect the changes; existing attached records are unaffected
Edit the field values of a record attached to a selected object (see “Viewing and Editing Object Records” on page 266)	Changes to field values affect the selected object only; the record fields cannot be changed. Existing and future objects with the record attached use the default values.
Edit the record field values of a symbol definition (see “Editing Symbol Default Record Values” on page 267)	Changes to field values affect future instances of the symbol. Existing symbols, other symbols with the record attached, and the record format resource are unaffected.

Alternatively, make global changes with the symbol utilities (see “Changing One Record Format Field” on page 274 and “Changing All Record Format Fields” on page 274).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Default Record Formats
Viewing and Editing Object Records
Editing Symbol Default Record Values
Creating Record Formats
Editing Record Formats
Splitting and Merging Record Formats

Linking Text to Record Formats
Database Connectivity

Editing Default Record Formats

A record format's fields and default values can be edited from the Resource Browser.

To edit record format values:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.

2. Select the record format.
3. Select **Resources > Edit**.

   The Edit Record Format dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Displays the name of the current record format
Field list	Lists the fields of the current record format
New	Creates a new field (as described in "Creating Record Formats" on page 262)
Edit	Edits the selected field
Remove	Deletes the selected field

4. Select the record format field to edit and click **Edit** to change the default value, or click **New** or **Remove** to add or delete fields.
5. Click **OK**.

   The edits do not affect existing values for attached records. Changes are applied to the default field values when attached to symbols or objects, or for future symbols with the record attached.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Record Formats
Editing Record Formats

Viewing and Editing Object Records

The Object Info palette Data tab is used to attach, view, and edit the record values for specific instances.

If selecting multiple objects with different records attached, the Data tab displays records that are attached to objects with a grayed box, but does not indicate which records are attached to which objects.

Resize the list boxes by selecting and dragging the resize bars between them.

To edit record field values for a selected object:

1. Select **Window > Palettes > Object Info**.

   The Object Info palette opens.

2. Select the object.

   Click the Data tab on the Object Info palette. The Object Info palette displays information about the record format(s) attached to the selected object(s).

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Name	Displays the name, if any, given to an object
Record Formats	Displays all records contained in the drawing; records attached to the selected object are indicated with an X in the check box to the left of the record name. If more than one object is selected, only the records attached to all objects display an X.
Record Fields	Displays all the fields in the selected record; if a default value was assigned to the field, it displays after the field name. If no item is currently selected in the drawing, the label **Record Field Defaults** displays instead.
Record Info	Edits the field values for the selected record; all entered values override any default values for the object

3. Select the desired record in the **Record Formats**, and then the desired field in the **Record Fields**. Change the **Record Info** the default value.

   This information applies to the selected object only; other new and existing objects with that record attached will still use the original (default) record format values. If a selected chair has a different part number from the default value that was entered in the record format, enter the new part number for that chair only. If the part number for all chairs has changed, edit the default record format instead.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Default Record Formats
Editing Symbol Default Record Values
Creating Record Formats
Editing Record Formats

Editing Symbol Default Record Values

The default values of a record attached to a symbol can be changed while editing a symbol definition (see “Editing Symbol Definitions” on page 247). This does not change the record format resource; if other symbols or objects have the record attached, they are unaffected. It does change the future default record values for that symbol.

To change the default field values of a symbol’s record format:

1. Select a symbol definition.
2. Select **Resources > Edit**.
3. Select the symbol component to edit, and click **Edit**.
4. In the Edit Symbol window, deselect all by clicking in an empty area of the window.
5. In the Object Info palette, click on the Data tab.

   When no items are selected, the Data tab displays *SYMBOL DEFAULTS* at the top.
6. Select the desired record in the **Record Formats**, and then the desired field in the **Record Fields**. Change the **Record Info** default value.
7. Click **Exit Symbol** at the top right of the drawing window to return to the drawing.

   The default field values of the record format for that symbol definition have been changed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Default Record Formats
Creating Record Formats
Editing Record Formats

Splitting and Merging Record Formats

Record format data can be manipulated, so that data from one record format can be split into two formats, or data from two record formats can be combined into one. No record data is lost during these transfers.

Splitting Record Formats

Record fields, and associated data, from one record format can be transferred to another new record format. This is useful, for example, when importing GIS data (Vectorworks Design Series required) with a significant number of record fields that may need to be reorganized.

To split a record format into two formats:

1. At least one record format must be present in the file. Select **Tools > Records > Split Record Format**.

   The Split Record Format dialog box opens. Select the record format to edit from the **Source Record Format** list.

Click to show/hide the parameters.

Parameter	Description
Source Record Format	Lists the record formats in the file; select the record format to edit
Fields to split	Lists the record fields in the selected source format; click in the first column to select fields to be split out
New Record Format name	Enter the name of the new record format to create. The split record fields are placed in the new record format.

2. Click to place a check mark next to the record fields that should be split from the selected format.
3. Enter a name for the new record format; it will contain the split record fields.
4. Click **OK**.

   The indicated record fields (and associated data) are removed from the source record format and placed in the new record format, in record format order. All objects which had the original record format attached now have both the original record format and the new record format attached. No record fields and data are lost during this operation.

Merging Record Formats

The record fields, and associated data, from two record formats can be merged into one record format.

To merge two record formats:

1. At least two record formats must be present in the file. Select **Tools > Records > Merge Record Formats**.

   The Merge Record Formats dialog box opens.

2. Select the format to be merged from the **Merge Record Format** list.

   The record fields from this record format will be merged, and this record format will be deleted.

3. Select the record format to receive the record fields that are merged.
4. Click **OK**.

   The merged record fields are appended to the end of the receiving format's record fields, in record format order. If there is a naming conflict and there are two fields with the same name, the transferred field name begins with an "_" (underscore). Any objects which had the first format attached now have the merged record attached instead. Any objects which already had the merged record attached now have the additional record fields included. The first Merge Record Format is deleted.

Creating Record Formats

Editing Record Formats

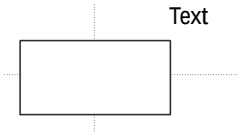
Linking Text to Record Formats

The **Link Text to Record** command links the text within a symbol definition to a field of the attached record. This is particularly useful for labeling symbols in a drawing with unique information, such as a part list number or price. To use this command, symbols and record formats must already exist in the drawing file. The link is made within a symbol's definition.

To link text to records within a symbol definition:

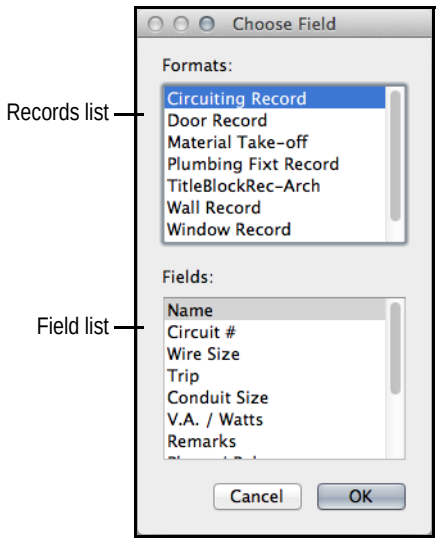
1. Edit the 2D symbol definition as described in "Editing Symbol Definitions" on page 247.
2. In the Edit Symbol window with nothing selected, create a line or block of text.

   Ensure the text is formatted with the desired font and style. The actual text is not important at this point. If desired, create and assign a class to the text.



3. Select the text and place it in the exact location where the record data value is to display.
4. With the text still selected, select **Tools > Records > Link Text to Record**.

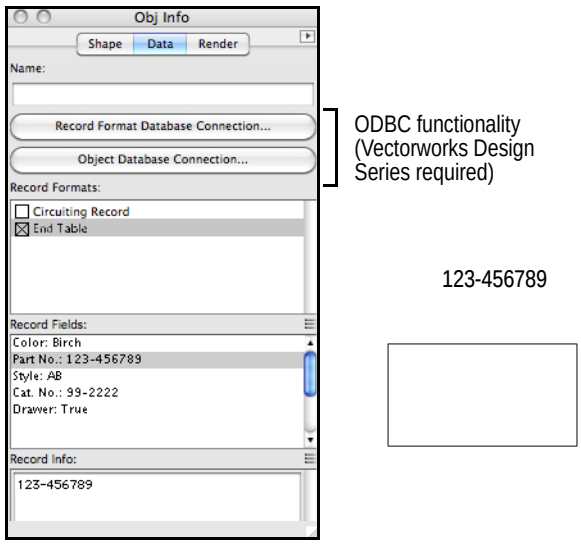
   The Choose Field dialog box opens.



5. Select the record format from the **Records** list.
6. In the list of fields, select the field of the selected record to associate with the text.
7. Click **OK**.

   Vectorworks adds the text object to the symbol definition, as well as all existing instances on the drawing. The default value of the selected field replaces the "dummy" text.
8. Click **Exit Symbol** at the top right of the drawing window to return to the drawing.

   Place the symbol on the drawing. The text linked to the record displays the field information. If necessary, to edit the value for that particular instance, select the field where the text was assigned in the Object Info palette Data tab. In the Edit Field box, enter the text to display in the symbol; the linked field information displays on the attached symbol.



Creating Record Formats
Attaching Record Formats to Symbols and Objects
Editing Record Formats
Database Connectivity
Modifying Objects by Record Value

Modifying Objects by Record Value

This command modifies the color, size, or height of objects in a drawing based on the numeric value of a particular record field attached to the selected items. This can be useful for analyzing and comparing objects in a drawing.

To modify objects by record value:

1. Select **Tools > Records > Modify by Record**.

   The Modify by Record Value dialog box opens.

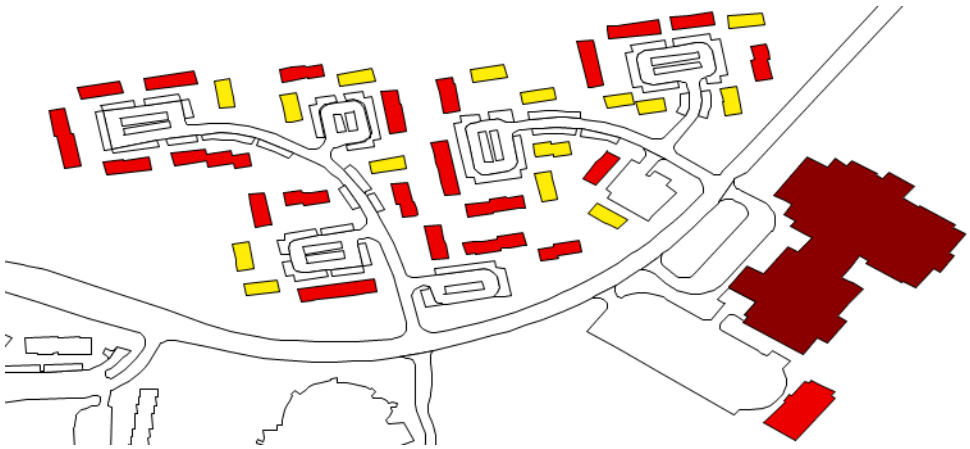
Click to show/hide the parameters.

Parameter	Description
Saved Settings Options	
Settings	Select <Active Settings> to use the settings currently selected in the dialog box; to use a set of saved settings, select them from the pull-down list
Save	Opens a dialog box to name and save the currently selected settings so that they can quickly be selected as a set
Manage	Opens the Manage Saved Sets dialog box to rename or delete sets of saved settings (see “Managing Saved Settings” on page 272)
Record	Lists the record formats available for the applicable objects in the current drawing (based on the settings in the **Apply to** and **Include components of** fields). Select the record that contains the field you want to use.
Record Field	Lists the fields available in the selected **Record** format; select the field to use to modify the objects

Parameter	Description
Color Gradient	Applies colors to the selected objects according to the numeric value in the **Record Field**. Use the fields in the Options box to select a gradient and enter a numeric range. The colors are automatically drawn from the gradient and assigned at equal intervals in the given range.
Color Gradient Options	
Apply to	Select the object attributes to which the specified colors will be applied: fill background or foreground, or pen background or foreground
Lower bound/Upper bound	Objects with a field value at or below the lower bound will be given the color at the start (left side) of the gradient. Objects with a value at or above the upper bound will be given the color at the end (right side) of the gradient. Objects with a value between the bounds will be given a color along the gradient.
Gradient	Select the desired gradient from either the default content or the current file's content (see "Resource Libraries" on page 219). The object colors will be taken from this gradient.
Color Ranges	Applies colors to the selected objects according to the numeric value in the **Record Field**. Use the fields in the Options box to define each numeric range and the color to be used.
Color Range Options	
Apply to	Select the object attributes to which the specified colors will be applied: fill background or foreground, or pen background or foreground
Ranges list	Lists the numeric ranges and associated colors that will be used to modify objects
Add/Edit	To add a range, click **Add**. To edit a range, select a range from the list and click **Edit** (or double-click a value). The Edit Color Range dialog box opens. Select a color, and then specify the **Lower Bound** and **Upper Bound** of field values to which the color will be applied. Click **OK**, and the range displays in the **Ranges list**.
Delete	Select a range from the list and click **Delete** to remove it from the list
Color Value	Applies colors to the selected objects according to the numeric value in the **Record Field**. Use the fields in the Options box to specify a color for each numeric value found among the selected objects.
Color Value Options	
Apply to	Select the object attributes to which the specified colors will be applied: fill background or foreground, or pen background or foreground
Values list	Lists the numeric values of the selected field for the selected objects, and the associated colors that will be used to modify the objects
Edit	Select a record field value from the list and click **Edit** (or double-click a value) to open the Edit Color Value dialog box. To apply a color to objects with this value, select the **Apply color** option, and then select a color from the pull-down list. Click **OK**, and the **Values list** is updated. To remove the color from objects with this value, access the Edit Color Value dialog box and deselect the **Apply color** option, Alternatively, deselect the **Apply** column on the **Values list**.
Resize	Scales the selected objects by a specific amount; in the Options box, enter the **Multiplier** to apply to the value in the **Record Field**

Parameter	Description
Extrude	Extrudes the selected objects by a specific amount; in the Options box, enter a **Multiplier** to apply to the value in the **Record Field**
Elevate 2D Polys	Converts 2D polygons or polylines to 3D polygons, using the elevation value in the record attached to each polygon/polyline to determine its equivalent 3D polygon elevation. Normally, site model source data from external sources, such as GIS, contain elevation data in attached records, so this is an automated way of creating the 3D contours to use as the basis of a site model. If needed, specify a **Multiplier** value other than 1; the elevation value in the attached record is multiplied by the specified factor to determine the contour elevation. Some GIS data is grouped. Ungroup the data, and select the option to **Attach group records to the resultant ungrouped objects**, so that each contour has individual data attached.
Apply to	Select whether to apply the attributes to the entire drawing, or to the currently selected objects only
Include components of	Optionally, select to apply the attributes to objects within groups and symbols, in addition to other objects. Groups and symbols are ignored if these checkboxes are not selected.

2. Click **OK**.
3. The color, size, or height of the drawing objects are modified.



This map of buildings and roads was modified according to the area value in each building's record. The smallest areas have a yellow fill, medium-sized areas have a red fill, and the largest area has a brown fill.

Managing Saved Settings

If you have saved settings to use with the **Modify by Record** command, use the **Manage** button on the Modify by Record Value dialog box to rename or delete these saved settings when needed.

To manage saved settings for the **Modify by Record** command:

1. Select **Tools > Records > Modify by Record**.

   The Modify by Record Value dialog box opens.

2. Click **Manage**.

   The Manage Saved Sets dialog box opens. Select the set to change.

Click to show/hide the parameters.

Parameter	Description
Rename	Select and enter a new name for the saved set.
Delete	Select to delete the set. You are prompted to confirm that you want to delete the saved set.

3. Click **OK** to close the Manage Saved Sets dialog box. Click **OK** again to close the Modify by Record Value dialog box.

Global Symbol Commands

The symbol commands manipulate record data attached to symbols in libraries and perform global data changes. Use these commands to change the default record format values of the field(s) attached to multiple symbol definitions, either after importing a file from a different source or during normal project changes.

The symbol utilities support the following functions:

Attaching Records

This command attaches the selected record format to all of the symbol definitions in a specified symbol folder.

To attach a record:

1. Select **Tools > Records > Attach Record**.

   The Attach Record dialog box opens.

2. From the **Symbol Folder** list, select the criteria for attaching a record.

   Select None to attach the record to the symbol definition(s) at the root of the symbol library (symbol definitions not in any folder). Select All to attach the record format to all of the symbol definition(s) in the file's symbol library. Selecting a symbol folder changes all of the symbol definitions only in that folder and any subfolders.

3. From the **Record Format** list, select one of the record formats defined in the current file to be attached to the selected symbol definition(s).

4. Click **OK**.

   Confirm the operation and the number of symbol definitions affected.

   Symbol instances already on the drawing are not affected.

5. Click **OK**.

   To quickly verify that a record has been attached to the symbol library, create a report. See "Creating Reports" on page 1316.

Detaching Records

This command detaches the selected record format from all symbol definitions in a selected symbol folder.

To detach a record:

1. Select **Tools > Records > Detach Record**.

   The Detach Record dialog box opens.

2. From the **Symbol Folder** list, select None, All, or a symbol folder, if any.

   Select None to detach the record from the symbol definition(s) at the root of the symbol library (symbol definitions not in any folder). Select All to detach the record format from all of the symbol definition(s) in the file's symbol library. Selecting a symbol folder changes all of the symbol definitions only in that folder and any subfolders.

3. From the **Record Format** list, select one of the record formats defined in the current file to detach from the symbol library.

4. Click **OK**.

   Confirm the operation and the number of symbol definitions affected.

Changing One Record Format Field

This command changes a specified field default value for a selected record format attached to symbol definitions. Symbols already placed on the drawing are unaffected.

To change one record format field:

1. Select **Tools > Records > Change One Field**.

   The Change One Field dialog box opens.

2. From the **Symbol Folder** list, select None, All, or a symbol folder, if any.

   Select None to change the record of the symbol definition(s) at the root of the symbol library (symbol definitions not in any folder). Select All to change the record format of all of the symbol definition(s) in the file's symbol library. Selecting a symbol folder changes all of the symbol definitions only in that folder and any subfolders.

3. From the **Record Format** list, select the record format.

   The **Field Name** selections depend on the record format selected.

4. Select the **Field Name** to change.

5. Enter the **New Value**.

6. Click **OK**.

   Confirm the operation and the number of symbol definitions affected.

Changing All Record Format Fields

This command changes several or all default field values for a selected record format attached to symbol definitions. Symbols already placed on the drawing are unaffected.

To change several or all record format fields:

1. Select **Tools > Records > Change All Fields**.

   The Change All Fields dialog box opens.

2. From the **Symbol Folder** list, select None, All, or a symbol folder, if any.

   Select None to change the record of the symbol definition(s) at the root of the symbol library (symbol definitions not in any folder). Select All to change the record format of all of the symbol definition(s) in the file's symbol library. Selecting a symbol folder changes all of the symbol definitions only in that folder and any subfolders.

3. From the **Record Format** list, select the record format.

4. Click **OK**.

   The Change Fields dialog box opens. The title bar displays the name of the record being edited.

5. Select the field name(s) to change and enter the new information.

   All of the field names of the record format are listed. If the record file has more than 16 fields, click **Next** to continue viewing the remaining fields in the record.

6. After making the desired changes, click **OK**.

   Confirm the operation and the number of symbol definition(s) affected.

Changing Symbol Attributes

This command assigns all of the objects within the selected symbol definitions to the specified class, and designates the objects to use any or all of the class attributes.

To change symbol attributes:

1. Select **Tools > Utilities > Change Symbol Attrs**.

   The Change Symbol Attributes dialog box opens.

2. From the **Symbol Folder** list, select None, All, or a symbol folder, if any.

   Select **None** to change the symbol attributes at the root of the symbol library (symbol definitions not in any folder). Select All to change the symbol attributes of all of the symbol definition(s) in the file's symbol library. Selecting a symbol folder changes all of the symbol definitions only in that folder and any subfolders.

3. Select the **Class** to assign the symbol definition.

4. Select the class attributes to use, and indicate how to treat class attributes that are not used.

   Either keep the object's existing attributes, or use the current defaults if a class attribute is not specified.

5. Click **OK**.

   Confirm the operation and the number of symbol definition(s) affected.

Listing Symbols and Folders

This command creates a text file list all of the symbol definitions and symbol folders within the active drawing.

To list symbols and folders:

1. Select **Tools > Utilities > List Symbols**.

   The List Symbol dialog box opens.

2. Select either **List Symbols and Folders** or **List Folders Only**.

3. Click **OK**.

   The Save File dialog box opens.

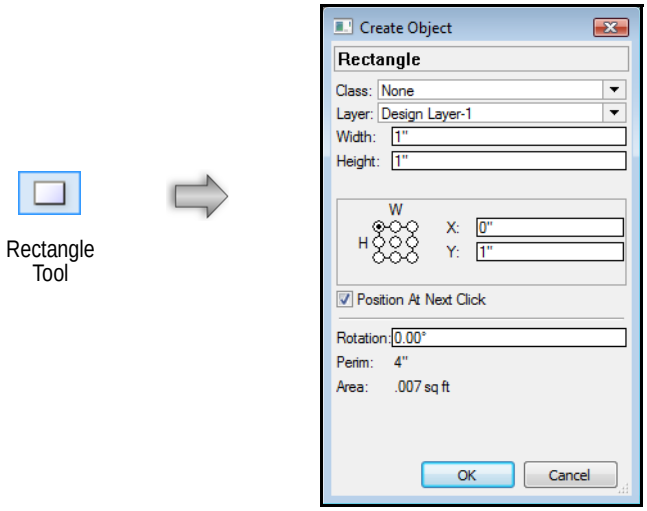
4. Enter a new file name or use the default name, and then click **Save**.

   A text file is created listing the names of all the folders and, if chosen, symbols within the current file.

Creating Shapes

Setting Parameters Before Creating Objects

Some of the creation tools can be used in conjunction with a properties or preferences dialog box. Not every tool includes this functionality.



To create objects and specify the properties before creation:

1. Double-click the desired tool (or press the tool's shortcut key twice).

   The tool's properties or preferences dialog box opens.
2. Enter the desired criteria.
3. Click **OK**.

   If **Position At Next Click** is selected, click the mouse to select the object's location in the drawing.

D Creating Objects from Shapes

The **Create Objects from Shapes** command uses an existing polyline or shape as the path for creating a variety of objects, as listed in the following table.

Object	Product
ceiling grids	Architect
guardrails	Architect, Landmark
handrails	Architect, Landmark
hardscape objects	Landmark
landscape areas	Landmark
landscape walls	Landmark
lighting pipe	Spotlight
linear material details	Architect, Landmark
massing models	Architect, Landmark
parking area	Landmark
parking along path	Landmark

Object	Product
piping runs	Architect
plants	Landmark
property lines	Architect, Landmark
repetitive unit details	Architect, Landmark
revision clouds	Architect, Landmark, Spotlight
roadways	Architect, Landmark
roadway (custom curb)	Architect, Landmark
roadways (NURBS)	Architect, Landmark
roadway (poly)	Architect, Landmark
seating layouts	Architect, Spotlight
site modifiers	Architect, Landmark
slabs	Architect
soft goods	Spotlight
spaces	Architect
stipple objects	Architect, Landmark, Spotlight
walls	Architect, Landmark, Spotlight

In addition to these standard objects, the **Create Objects from Shapes** command supports custom path objects (custom path plug-in objects with a .vso extension). For details, see “Creating Custom Path Objects” on page 1789.

Walls created with the **Create Objects from Shapes** command use the current settings made active by the **Wall** tool (Vectorworks Architect or Landmark required), and slabs use the current **Slab** tool settings (Architect required).

In the Vectorworks Fundamentals workspace, walls can be created from polygons using the **Create Walls from Polygon** command (see Creating Walls from a Polygon).

To create an object from a shape:

1. Draw a polyline or select an existing shape such as a line, rectangle, oval, polygon, arc/circle, rounded rectangle, polyline, or 3D polygon representing the path for creating the object.
2. With the item selected, select the **Create Objects from Shapes** command from the appropriate menu:
   - Architect workspace: **AEC > Create Objects from Shapes**
   - Landmark workspace: **Landmark > Create Objects from Shapes**
   - Spotlight workspace: **Modify > Create Objects from Shapes**

   Alternatively, right-click (Windows) or Ctrl-click (Mac) and select **Create Objects from Shapes** from the context menu.

   The Create Objects from Shapes dialog box opens.
3. Select the type of object and the creation parameters.

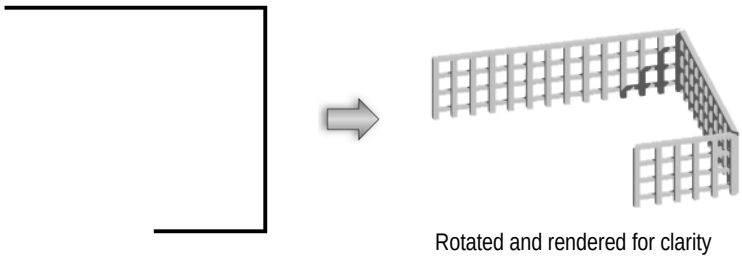
Click to show/hide the parameters.

Parameter	Description
Object Type	Select the type of object to create based on the shape path; the parameters available vary based on the object type selected
Offset	For guardrails, handrails, landscape walls, some roadway types, and walls, select whether to offset the object to the left, right, or centered on the shape; select **Custom** to enter a custom offset distance
Custom Offset	For guardrails, handrails, landscape walls, some roadway types, and walls, specify the custom offset distance from the object, if any
Show Properties Dialog	Select to display the Object Properties dialog box for the specified object type prior to creating the object; this field is not applicable for plant objects because the default place plant settings are used
Delete Source Shapes	Select to delete the original path or shape after drawing the object
Preserve Records (Vectorworks Architect or Landmark required)	If **Delete Source Shapes** is selected, select this option to apply record data from the source shape to the object that will be created
Continuous Spacing (Vectorworks Architect or Landmark required)	Select to automatically maintain continuous spacing between uprights from one object to the next based on the **Upright Spacing** value for guardrails or handrails

4. Click **OK**. If **Show Properties Dialog** was selected, the object properties dialog box for the specified object type opens. Enter the appropriate parameters, and then click **OK**.

   If you are creating guardrails or handrails and **Continuous Spacing** was selected in the Create Objects from Shapes dialog box, enter the distance between the first upright and the start of the shape in the **1st Upright Spacing** field of the object properties dialog box. If **Continuous Spacing** was not selected, this value is applied to the first upright at the beginning of each segment of the shape. Also enter the spacing to maintain between uprights in the **Upright Spacing** field of the object properties dialog box.

   The object is created from the shape and its parameters can be edited in the Object Info palette.



Rotated and rendered for clarity

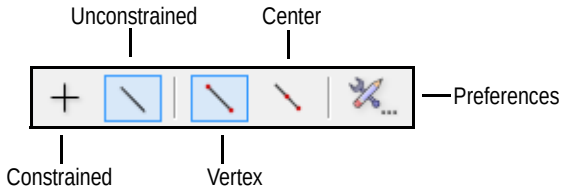
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Custom Path Objects

Creating Lines

Creating Single Lines

Use the **Line** tool to create single lines.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

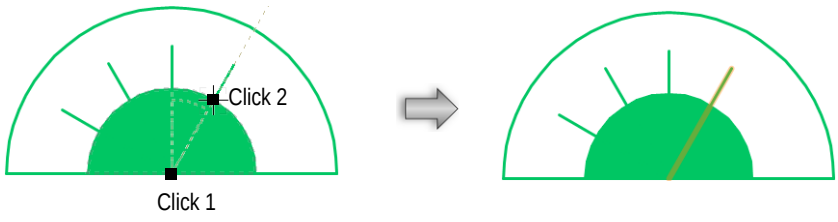


Mode	Description
Constrained	Constrains the line to be vertical, horizontal, and 30° or 45° from vertical or horizontal in any direction
Unconstrained	Draws the line at any angle Press and hold the Shift key to snap the line to predetermined angles
Vertex	The first click is the start point of the line
Center	The first click is the center point of the line; the line is drawn outward in both directions from the center point
Preferences	Scales the length of the line by a specific amount as it is drawn

Creating a Line from Start Point to End Point

To create a single line from its start point to its end point:

1. Click the **Line** tool from the Basic palette.
2. Click either the **Constrained** or **Unconstrained** mode.
3. To scale the line as you draw it, click **Preferences** from the Tool bar, and enter the scale factor (must be larger than zero). The created line will be the length of the line you draw multiplied by this number. For example, enter 2 to create a line twice as long as the drawn line, or enter .5 to create a line half as long as the drawn line. By default, the value is 1, which does not affect the line length.
4. Click **Vertex** mode.
5. Click at the line's start point.
6. Click again to set the end point of the line. If you entered a scale factor other than 1, the length of the finished line is adjusted accordingly.



With the Length Scale Factor set to 1.5, click the center of the filled arc to start the line, and then click a point on the edge of the filled arc

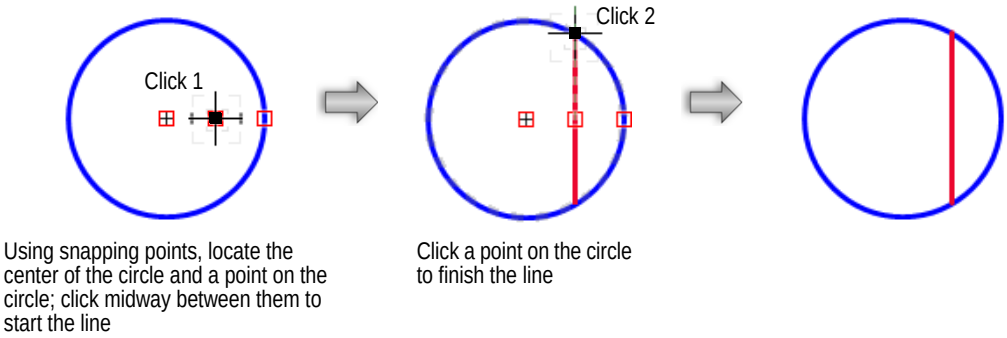
The finished line is 1.5 times longer than the radius of the filled arc

Creating a Line from the Center Point

To create a single line from its center point:

1. Click the **Line** tool from the Basic palette.

2. Click either the **Constrained** or **Unconstrained** mode.
3. To scale the line as you draw it, click **Preferences** from the Tool bar, and enter the scale factor (must be larger than zero). The created line will be the length of the line you draw multiplied by this number. For example, enter 2 to create a line twice as long as the drawn line, or enter .5 to create a line half as long as the drawn line. By default, the value is 1, which does not affect the line length.
4. Click **Center** mode.
5. Click at the line's center point.
6. Click again to set one of the line's end points. If you entered a scale factor other than 1, the length of the finished line is adjusted accordingly.

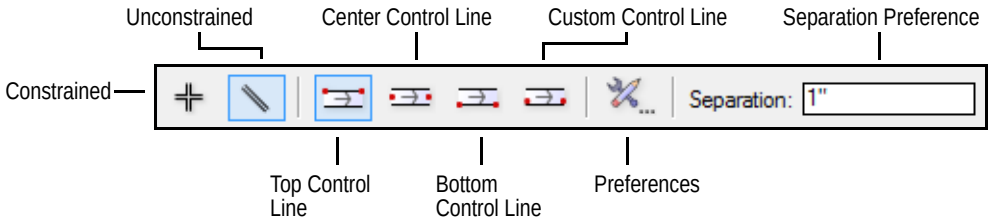


Click here for a video tip on this topic (Internet access required).

Creating Double Lines
Creating a Break Line
Displaying and Reversing Object Direction
Marker Attributes

Creating Double Lines

The **Double Line** tool creates a wide variety of constrained and unconstrained double lines. Set the width between the double lines, the offset from the cursor, and whether to create components between the double lines.



To create double lines:

1. Click the **Double Line** tool from the Basic palette.
2. Click either the **Constrained** or **Unconstrained** mode.

   Constrained lines are drawn at 30°, 45°, and 90° angles, and their complements are drawn in increments of 30°, and 45°.

   Unconstrained lines can be drawn at any angle.

   Press and hold the Shift key while drawing a line in Unconstrained mode to snap the line to predetermined angles.
3. Click the desired **Offset** mode to specify the offset method.

Offset Mode	Description
Top Control Line	The cursor draws the top line
Center Control Line	Lines are equidistant from the line drawn by the cursor
Bottom Control Line	The cursor draws the bottom line
Custom Control Line	Specify an offset value from the line drawn by the cursor

4. Enter the distance between the double lines in the **Separation** field on the Tool bar.
5. Click the **Preferences** button to set other parameters.

Click to show/hide the parameters.

Parameter	Description
Separation	Enter the distance between the double lines; same as the **Separation** field on the Tool bar
Control Offset	For Custom Control line mode, enter the distance from the top line of the cursor. The distance must be less than the **Separation** distance between the double lines.
Options	
Create Lines	Creates parallel lines
Create Polygons	Creates a double-line polygon with closed ends
Create Lines and Polygons	Creates parallel lines with a polygon between them; the polygon has a line weight of zero
Components	Click to define components between the double lines

6. Click **OK** to accept the changes.
7. Click at the double line's start point.
8. Click at the double line's end point.

Click here for a video tip on this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Single Lines
Creating a Break Line
Applying Components Between Double Lines
Displaying and Reversing Object Direction
Marker Attributes

Applying Components Between Double Lines

To apply a component between the double lines:

1. In the Double Line Preferences dialog box, click **Components**.

   The Components dialog box opens. Specify the component details.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Preview	Displays a preview of the components between the double lines, including the defined components; the preview is drawn from left to right, so the "top" of the preview, by default, indicates the left part of the double lines as they will be drawn. The arrow shows the drawing direction.
Overall Thickness	The thickness of the double line with components is defined by the sum of the component thicknesses
Components	Lists the components that form the structure of the double line, in order from left to right as displayed in the preview. To change the order of a component, click and drag within the # column. The Core designation applies to walls, but not to double lines.
New	Click to define the components between the double lines; see "Creating Wall Components" on page 513
Edit	Opens the Wall Component Settings dialog box to edit the selected component's thickness and attributes (you can also double-click on a component to open the Wall Component Settings dialog box)
Delete	Deletes the selected component; the double line thickness is adjusted accordingly

2. When the components have been defined, click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Double Lines

Creating a Break Line

The **Break Line** tool creates one of three types of break lines: straight, curved, or arc.

 To draw a break line:

1. Click the **Break Line** tool from the Dims/Notes tool set.
2. Click in the drawing to place the break line and move the mouse to indicate the line length. Click again to set the end of the break line.

   If this is the first break line placed in this session, the Break Line Object Properties dialog box opens. The settings displayed apply to all break lines created during this session and can be edited in the Object Info palette after placement.
3. Click **OK**.

4. The break line parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

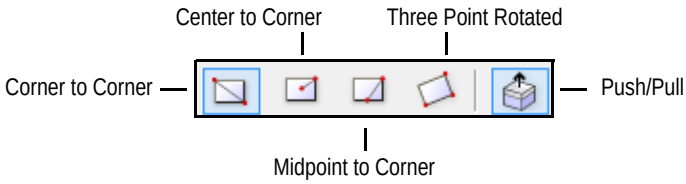
Parameter	Description
Break Style	Selects the style of the break line (Straight, Curved, or Arc)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Break Width	Indicates the width of the break only
Break Height	Indicates the height of the break only
Break Radius	Sets the radius of the break only
Number of Breaks	Indicates whether a single or multiple break should be drawn

Creating Single Lines
Creating Double Lines

Creating Rectangles

The **Rectangle** tool can be used to create rectangular shapes and has four modes available. The **Push/Pull** toggle mode is available in 3D views for instantly extruding the rectangle after creation.



Mode	Description
Corner to Corner	Defines the rectangle by clicking two diagonal points
Center to Corner	Defines the rectangle by the distance from the center to one corner
Midpoint to Corner	Defines the rectangle by the distance from the midpoint of one side to an opposite corner
Three Point Rotated	Creates a rectangle that is rotated by a specified angle
Push/Pull (3D views only)	Instantly extrudes the rectangle after creation; available in 3D views

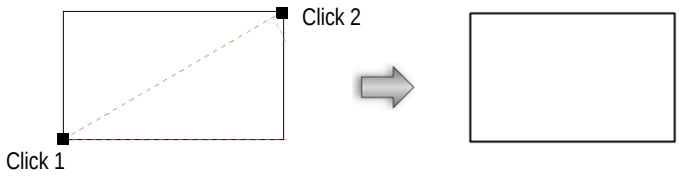
To create a square (1:1 rectangle), press and hold the Shift key while drawing. To create a rectangle with golden ratio proportions (approximately 1:1.618), press and hold Ctrl+Shift (Windows) or Cmd+Shift (Mac) while drawing.

The **Ratio** field in the Object Info palette displays square, golden, or the ratio of the sides of an unconstrained rectangle.

Creating a Rectangle by Corner to Corner

To create a rectangle by corner to corner:

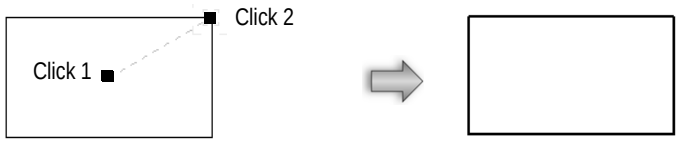
1. Click the **Rectangle** tool from the Basic palette and click **Corner to Corner** from the Tool bar.
2. Click at the rectangle's start point; this point becomes one corner of the rectangle. Move the mouse to the opposite corner until the desired size is previewed.
3. Click to set a corner point on the rectangle.



Creating a Rectangle by Center to Corner

To create a rectangle by center to corner:

1. Click the **Rectangle** tool from the Basic palette, and click **Center to Corner** from the Tool bar.
2. Click at the rectangle's start point; this point becomes the center of the rectangle. Move the mouse to an opposite corner until the desired size is previewed.
3. Click to set a corner point on the rectangle.

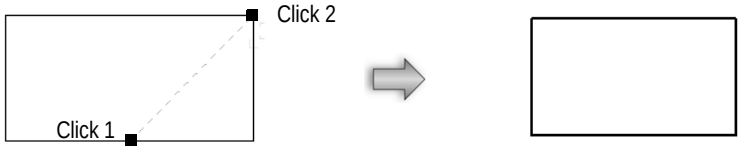


Creating a Rectangle by Midpoint to Corner

To create a rectangle by midpoint to corner:

1. Click the **Rectangle** tool from the Basic palette, and click **Midpoint to Corner** from the Tool bar.
2. Click at the rectangle's start point; this point becomes the center of one of the rectangle's sides. Move the mouse to an opposite corner until the desired size is previewed.

   Press the Alt key (Windows) or Option key (Mac) to orient the rectangle vertically.
3. Click to set a corner point on the rectangle.

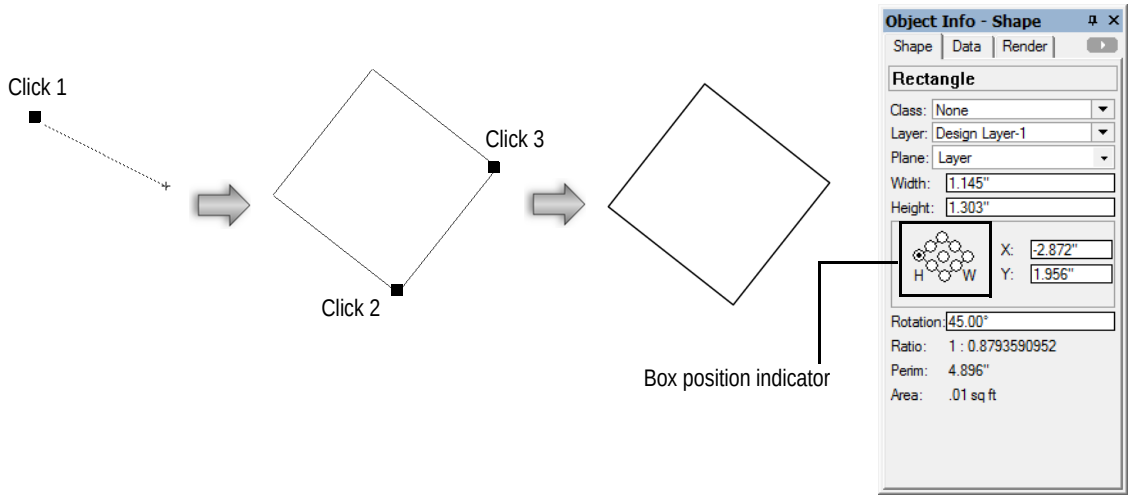


Creating a Three Point Rotated Rectangle

Rotated rectangles are rectangles that are rotated at a specified angle.

To create a three point rotated rectangle:

1. Click the **Rectangle** tool from the Basic palette, and click **Three Point Rotated** from the Tool bar.
2. Click to set the start of the rectangle. Move the mouse to set the rotated angle.
3. Click to set the rotation angle. Move the mouse until the desired size is previewed.
4. Click to create the three point rotated rectangle; the bounding box position indicator in the Object Info palette matches the rotation of the rotated rectangle. (If multiple rotated rectangles are selected, the box position indicator matches the rotation of one rectangle.)

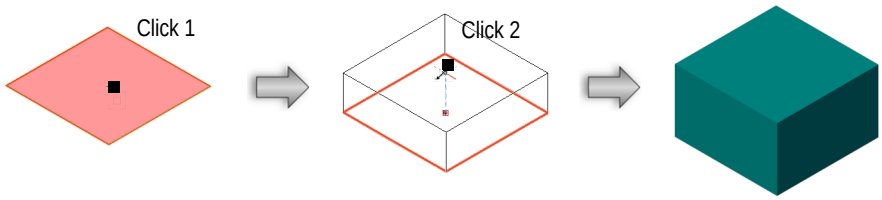


Creating Extruded Rectangles

A planar rectangle that has been created in a 3D view can be extruded immediately after creation.

To extrude a created rectangle:

1. Enable the Push/Pull toggle mode in the Tool bar for the **Rectangle** tool.
2. Create a rectangle using any of the modes described in the previous sections.
3. With the rectangle still selected, click, move the cursor, and then click to create the extrude.



A planar rectangle can be extruded at any time with the **Push/Pull** tool. See “Direct Modeling with the Push/Pull Tool” on page 357.

The extrude height can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), or modified in the Object Info palette.

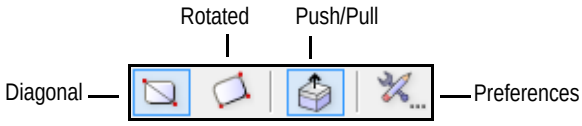
Creating Rounded Rectangles
Push/Pull Mode of Planar Tools

Creating Rounded Rectangles

Rounded rectangles can be created with symmetrical, proportional, or unconstrained corner diameters.

Symmetrical corners have the same X and Y rounded corner diameters. Proportional corners are created as one third of the width and height of the rounded rectangle; for example, a rounded rectangle with a width of 6” and a height of 3” has a Diam X value of 2” and a Diam Y value of 1”. Rounded rectangles with both symmetrical and proportional corners have the same horizontal and vertical arc lengths. Unconstrained rounded rectangles can also be created using specific X and Y corner diameters.

Two modes are available. The **Push/Pull** toggle mode is available in 3D views for instantly extruding the rectangle after creation.



Mode	Description
Diagonal	Defines the dimensions of the box containing the rounded rectangle
Rotated	Defines the width and height lengths to create the rounded rectangle, which can be rotated if desired
Push/Pull (3D views only)	Instantly extrudes the rounded rectangle after creation; available in 3D views

Rounded Rectangle by Box

To create rounded rectangles by box:

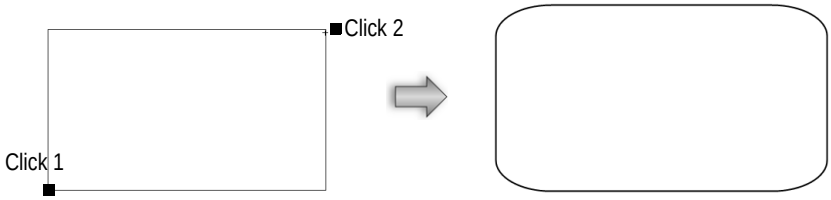
1. Click the **Rounded Rectangle** tool from the Basic palette, and click **Diagonal** from the Tool bar.
2. Click the **Preferences** button.

   The Round Rect Preferences dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Corner Styles	Select the method of rounding the rectangle corners • Symmetrical: the corner diameters are identical in the X and Y direction • Proportional: the corner diameters are one third of the rounded rectangle width and height • Unconstrained: no constraints on the X and Y corner diameters
Corner Diameters	For symmetrical or unconstrained rounded rectangle corner style, enter the corner X and corner Y diameter; symmetrical automatically keeps the same value for X and Y

3. Specify the preferences and click **OK**.
4. Click to set the start point.
5. Click to set the end point of the rectangle.

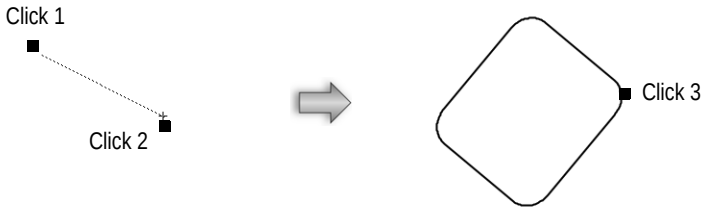


Rounded Rectangle by Width and Height

To create rounded rectangles by width and height:

1. Click the **Rounded Rectangle** tool from the Basic palette, and click **Rotated** from the Tool bar.

2. Set the rounded rectangle preferences as described in “Rounded Rectangle by Box” on page 287.
3. Click to set the start point.
4. Click to define the rounded rectangle rotation angle and width.
5. Click to define the rounded rectangle height.

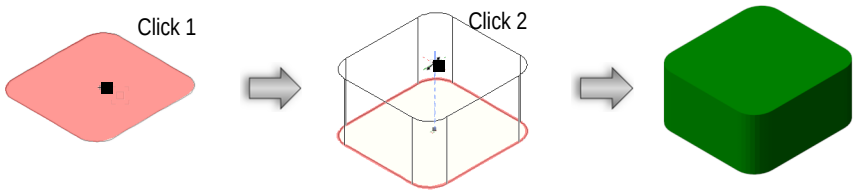


Creating Extruded Rounded Rectangles

A planar rounded rectangle that has been created in a 3D view can be extruded immediately after creation.

To extrude a created rounded rectangle:

1. Enable the Push/Pull toggle mode in the Tool bar for the **Rounded Rectangle** tool.
2. Create a rounded rectangle using any of the modes described in the previous sections.
3. With the rectangle still selected, click, move the cursor, and then click to create the extrude.



A planar rectangle can be extruded at any time with the **Push/Pull** tool. See “Direct Modeling with the Push/Pull Tool” on page 357.

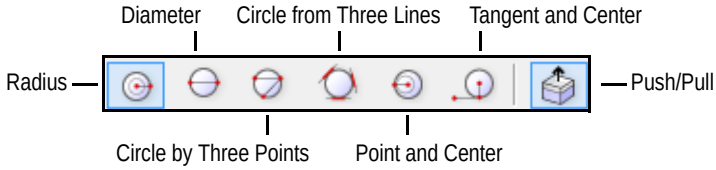
The extrude height can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), or modified in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Rectangles
Push/Pull Mode of Planar Tools

Creating Circles

The **Circle** tool has six modes and the **Push/Pull** toggle mode.



Mode	Description
Radius	Defines the circle by its radius
Diameter	Defines the circle by its diameter
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

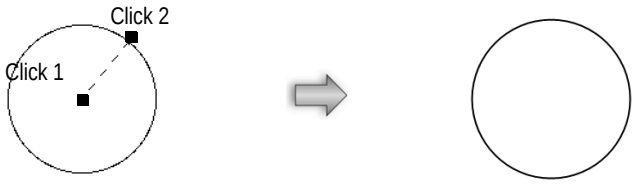
Mode	Description
Circle by Three Points	Defines the circle by its circumference
Circle from Three Lines	Defines the circle by making it tangent to two or three selected lines
Point and Center	Defines the circle by a point on its circumference and then its center
Tangent and Center	Defines the circle by defining a tangent to the circle and then its center
Push/Pull (3D views only)	Instantly extrudes the circle after creation; available in 3D views

To create a NURBS curve, draw a circle and then select **Modify > Convert > Convert to NURBS**.

Circle by Radius

To create a circle by radius:

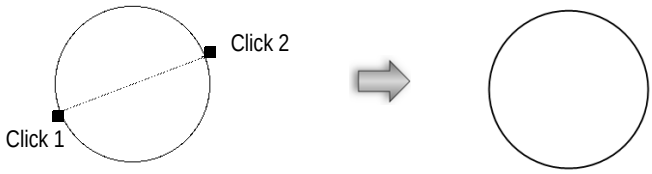
1. Click the **Circle** tool from the Basic palette, and select the **Radius** mode.
2. Click to set the center of the circle.
3. Move the mouse to the desired radius and click to set the radius of the circle.



Circle by Diameter

To create a circle by diameter:

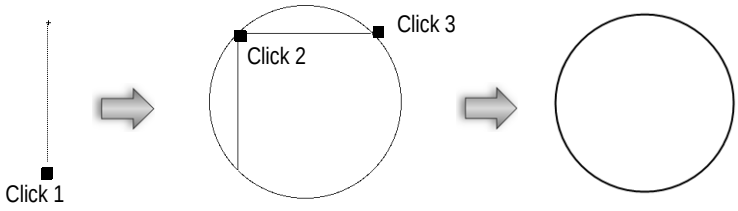
1. Click the **Circle** tool from the Basic palette, and select the **Diameter** mode.
2. Click to set the first point on the circle diameter.
3. Move the mouse to the desired diameter and click to set the diameter of the circle.



Circle by Three Points

To create a circle by three points:

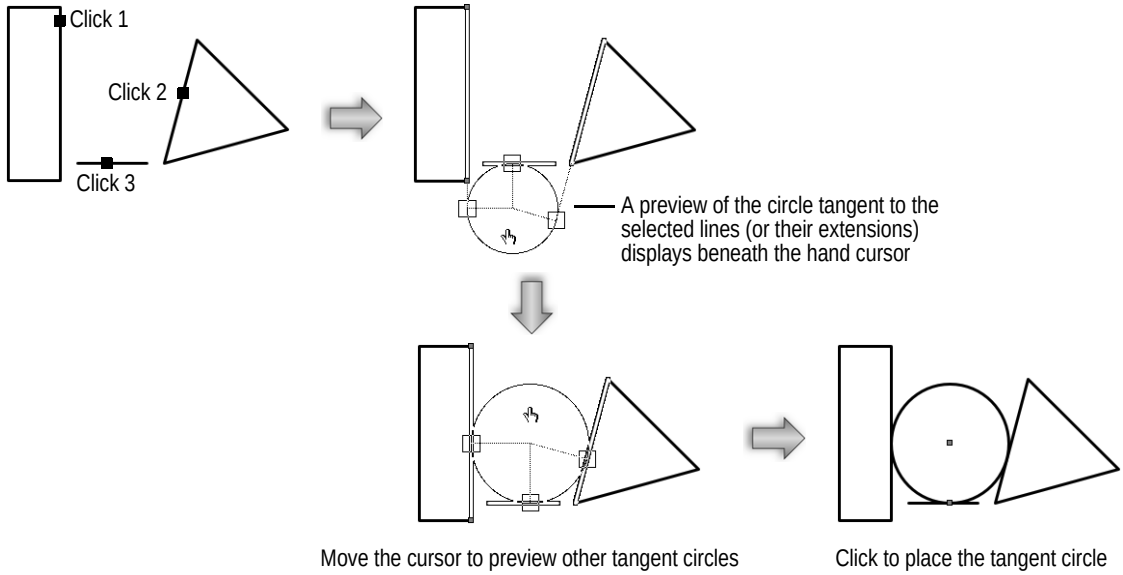
1. Click the **Circle** tool from the Basic palette, and select the **Circle by Three Points** mode.
2. Click to set the first point on the circle diameter.
3. Move the mouse and click to set the second point on the circle and click again to set the third point on the circle.



Circle from Three Lines

To create a circle tangent to three lines:

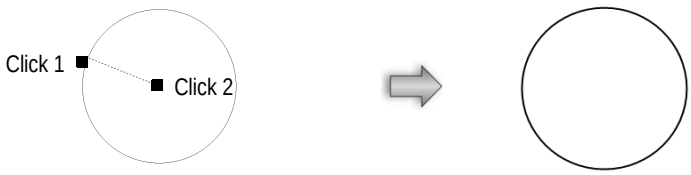
1. Click the **Circle** tool from the Basic palette, and select the **Circle from Three Lines** mode.
2. Click to select the three lines or linear segments to which the circle will be tangent. Each line is highlighted as it is selected. The lines (or their extensions) must intersect in at least two places.
3. When the third line is selected, a preview of a tangent circle displays where the cursor is currently located. You can create circles tangent to two or three lines. In places where only two lines intersect, the size of the circle is controlled by the cursor location. Move the cursor until the desired tangent circle displays, and then click to set the circle.



Circle by Point and Center

To create a circle by point and center:

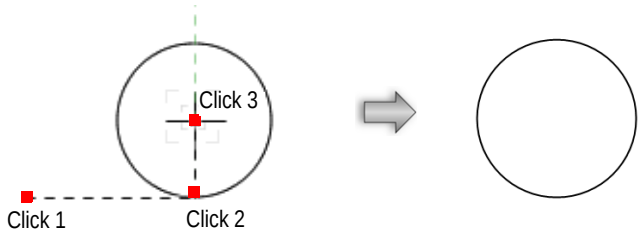
1. Click the **Circle** tool from the Basic palette, and select the **Point and Center** mode.
2. Click to set the first point on the circumference of the circle.
3. Move the mouse to the center of the circle and click to set the radius of the circle.



Circle by Tangent and Center

To create a circle by tangent and center:

1. Click the **Circle** tool from the Basic palette, and select the **Tangent and Center** mode.
2. Click to set the first point on the line tangent to the circle.
3. Click to set the second point on the line tangent to the circle, defining the tangent line.
4. Move the mouse to define the radius perpendicular to the tangent. Click to set the radius of the circle. It is not necessary to click on the center point; click anywhere along an axis parallel to the tangent. This allows you to set the radius based on other geometry, if desired.

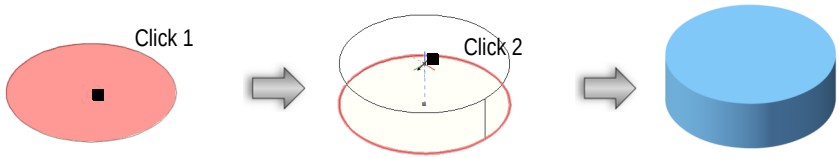


Creating Extruded Circles

A planar circle that has been created in a 3D view can be extruded immediately after creation, creating a cylinder.

To extrude a created circle:

1. Enable the Push/Pull toggle mode in the Tool bar for the **Circle** tool.
2. Create a circle using any of the modes in the Tool bar.
3. With the circle still selected, click, move the cursor, and then click to create the extrude.



A circle can be extruded at any time with the **Push/Pull** tool. See “Direct Modeling with the Push/Pull Tool” on page 357.

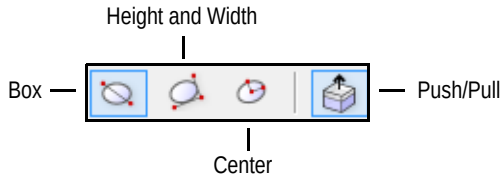
Cylinder height and radius can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), the radius can be modified in the Object Info palette, and the resolution can be adjusted through the **3D Conversion Resolution** setting (see “3D Preferences” on page 54).

Click here for a video tip on this topic (Internet access required).

Creating Circles
Push/Pull Mode of Planar Tools

Creating Ovals

The **Oval** tool has three modes available. The **Push/Pull** toggle mode is available in 3D views for instantly extruding the oval after creation.

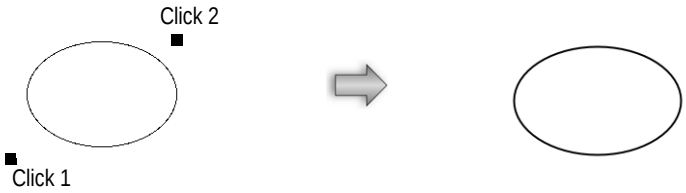


Mode	Description
Box	Defines the dimensions of the box containing the oval
Height and Width	Defines the height and width lengths to create the oval
Center	Defines the center, then the major and minor radii to create the oval
Push/Pull (3D views only)	Instantly extrudes the oval after creation; available in 3D views

Oval by Box

To create an oval by box:

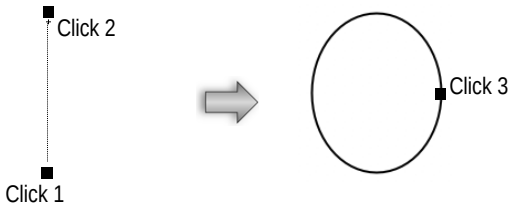
1. Click the **Oval** tool from the Basic palette and select the **Box** mode.
2. Click to set the first point of the box containing the oval and then click again to set.
   To create a true circle, press the Shift key while creating the oval.



Oval by Height and Width

To create an oval by width and height:

1. Click the **Oval** tool from the Basic palette and select the **Height and Width** mode.
2. Click to set the first point of the oval height, and then move the mouse to define the height.
3. Move the mouse, and then click to define the oval width.



Oval by Center

To create an oval by center:

1. Click the **Oval** tool from the Basic palette and select the **Center** mode.

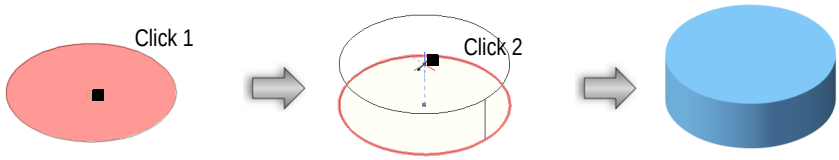
2. Click to set the center point of the oval, and then move the mouse and click to define one of the radii.
3. Move the mouse and then click to define the other radius.

Creating Extruded Ovals

A planar oval that has been created in a 3D view can be extruded immediately after creation.

To extrude a created oval:

1. Enable the Push/Pull toggle mode in the Tool bar for the **Oval** tool.
2. Create an oval using any of the modes in the Tool bar.
3. With the oval still selected, click, move the cursor, and then click to create the extrude.



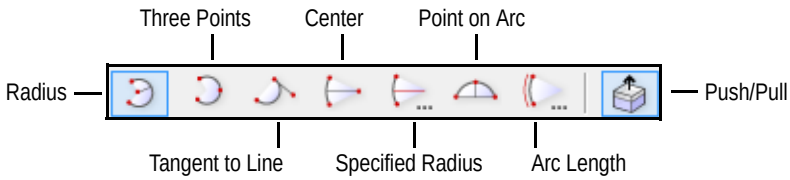
An oval can be extruded at any time with the **Push/Pull** tool. See “Direct Modeling with the Push/Pull Tool” on page 357.

The extrude height can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), or modified in the Object Info palette.

Creating Arcs

The **Arc** tool, which creates circular arcs of any angle, has seven modes. Create an arc by radius, three points, tangent, two points and center, two points and radius, two end points and another point on the arc, or by arc length and optionally, chord length. The **Push/Pull** toggle mode is available in 3D views for instantly extruding the arc after creation.

Click here for a video tip on this topic (Internet access required).



Mode	Description
Radius	Defines the arc by its center
Three Points	Defines the arc by clicking three points
Tangent to Line	Defines the arc by defining a tangent to a line
Center	Defines an arc by clicking two points and a center

Mode	Description
Specified Radius	Defines an arc by two points and a specified radius
Point on Arc	Defines an arc by two end points and another point on the arc
Arc Length	Defines an arc by the arc length and specified chord length
Push/Pull (3D views only)	Instantly extrudes the arc after creation; available in 3D views

Either move the mouse to draw the arc or use the Data bar to enter an angle. Degrees start on the positive X axis (the "East" direction), and increase going counter-clockwise. ("East" is 0 degrees; "North" is 90 degrees; and "West" is 180 degrees.) To draw a quarter-circle wall beginning at 0 degrees and ending at the 6 o'clock position, enter -90 (minus 90) degrees. Enter 270 to draw an arc three-quarters of a circle around.

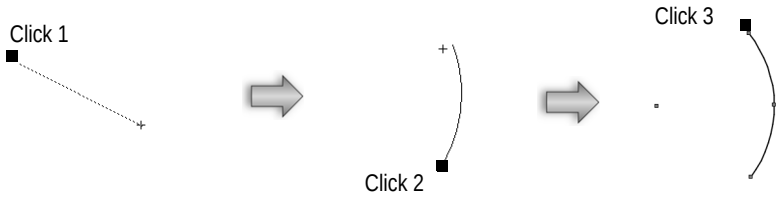
To create a NURBS curve, draw an arc and then select **Modify > Convert > Convert to NURBS**.

To edit an arc, click the middle handle with the **Selection** tool and drag to change the arc radius. Press the Option key (Mac) or Alt key (Windows) to change the arc's size.

Arc by Radius

To create an arc by its radius:

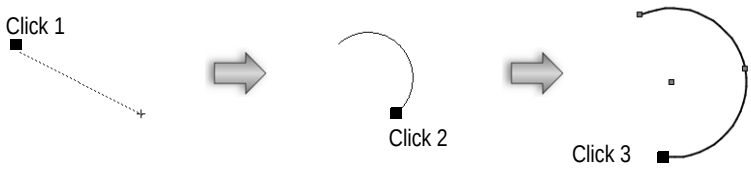
1. Click the **Arc** tool from the Basic palette, and select the **Radius** mode.
2. Click to set the center of the arc.
3. Click the start point of the arc. Move the mouse until the desired arc orientation and size is previewed.
4. Click to set the end point of the arc.



Arc by Three Points

To create an arc by three points:

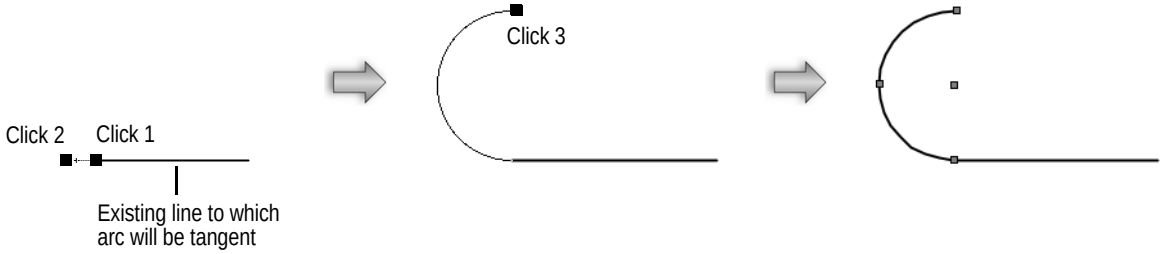
1. Click the **Arc** tool from the Basic palette, and select the **Three Points** mode.
2. Click to set the start point of the arc.
3. Click to set the point for the arc to pass through. Move the mouse until the desired arc orientation and size is previewed.
4. Click to set the end point of the arc.



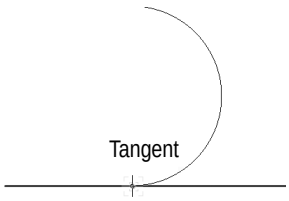
Arc Tangent to Line

To create an arc by tangent:

1. Click the **Arc** tool from the Basic palette, and select the **Tangent to Line** mode.
2. Click to set the start point of the arc.
3. Click to define the line to which the arc will be tangent. Move the mouse until the desired arc orientation and size is previewed.
4. Click to set the end point of the arc.



Use tangent snapping to assist with drawing the arc tangent to geometry; see “Snap to Tangent (2D Only)” on page 142.

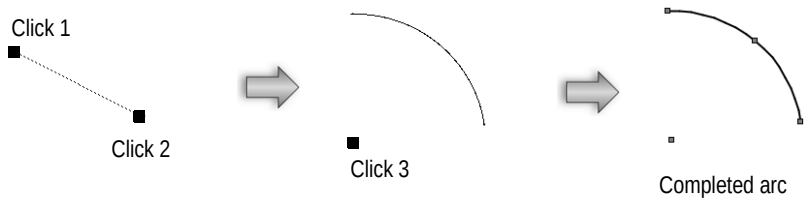


Arc by Center

To create an arc by two points and center:

1. Click the **Arc** tool from the Basic palette, and select the **Center** mode.
2. Click to set the start point of the arc.
3. Click to set the end point of the arc.

   As the cursor moves, the center of the arc is manipulated.
4. Click outside the arc to set the center.



Arc by Specified Radius

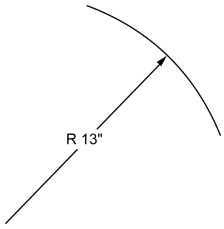
To create an arc by two points and a specified radius:

1. Click the **Arc** tool from the Basic palette, and select the **Specified Radius** mode.

2. Click to set the start point of the arc.
3. Click to set the end point of the arc.

   The Arc Radius dialog box opens. Enter the length of the radius.
4. Click **OK**.

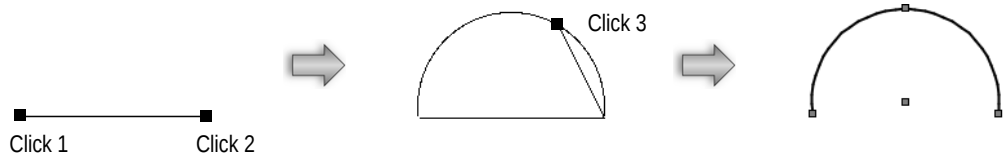
   The arc is created.



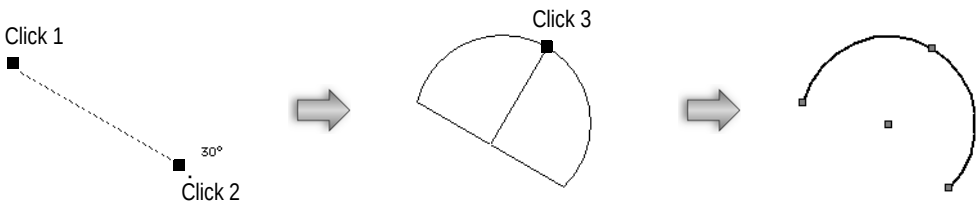
Arc by Point on Arc

To create an arc by two end points and another point on the arc:

1. Click the **Arc** tool from the Basic palette, and select the **Point on Arc** mode.
2. Click to set the start point of the arc.
3. Click to set the end point of the arc.
4. Move the mouse until the desired arc shape and size is previewed, and then click to set the arc. To constrain the angle of the line between the arc endpoints, press the Shift key when drawing the arc.



Without the Shift key, the angle of the line between the arc endpoints (1st and 2nd clicks) is unconstrained, and the chord defining the arc height is an unconstrained line drawn from the second endpoint



When the Shift key is pressed, the angle of the line between the arc endpoints (1st and 2nd clicks) is constrained, and the chord defining the arc height is perpendicular to the center of that line

Arc by Arc Length

To create an arc by arc length and chord length:

1. Click the **Arc** tool from the Basic palette, and select the **Arc Length** mode.
2. Click to set the start point of the arc.
3. Click to set the end point of the arc.

The Arc Length dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Arc Length	The drawn arc length is displayed, and can be changed; this allows you to create several arcs with the same chord length and varying arc lengths
Specify Chord Length	Select to manually specify the chord length, changing the arc endpoint
Chord Length	The chord length is the distance between the two end points of the arc; the arc length must be greater than the chord length

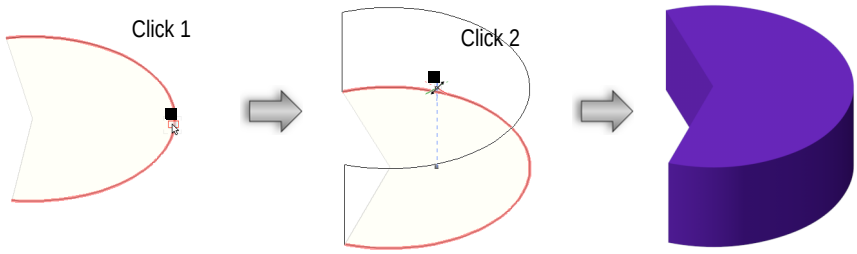
4. Enter the desired parameters, and click **OK** to create the arc.

Creating Extruded Arcs

A planar arc that has been created in a 3D view can be extruded immediately after creation.

To extrude a created arc:

1. Enable the Push/Pull toggle mode in the Tool bar for the **Arc** or **Quarter Arc** tool.
2. Create an arc using any of the modes in the Tool bar.
3. With the arc still selected, click, move the cursor, and then click to create the extrude.



An arc can be extruded at any time with the **Push/Pull** tool. See “Direct Modeling with the Push/Pull Tool” on page 357.

The extrude height and radius can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), the radius can be modified in the Object Info palette, and the resolution can be adjusted through the **3D Conversion Resolution** setting (see “3D Preferences” on page 54).

Creating Quarter Arcs

The **Quarter Arc** tool creates circular 90° arcs or 90° elliptical arc polyline objects. The **Push/Pull** toggle mode is available in 3D views for instantly extruding the arc after creation.

Quarter arcs are listed as polylines in the Object Info palette.

To create quarter arcs:

1. Click the **Quarter Arc** tool from the Basic palette.
2. Click at the start point for the arc.
3. Move the mouse until the desired arc shape and size is previewed, and then click to set the arc. To constrain the angle of the quarter arc to 45 degrees, press the Shift key when drawing the arc.

Creating Arcs
Creating Extruded Arcs
Marker Attributes

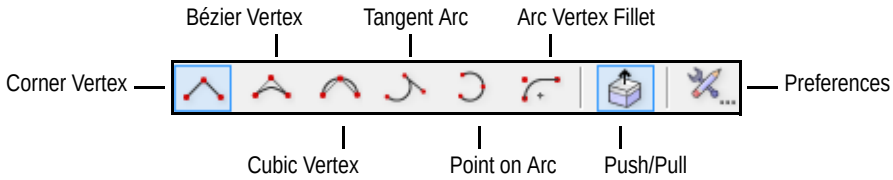
Creating Polylines

Polyline Tool

The **Polyline** tool creates open and closed polylines—objects made of a series of connected arcs, curves, or lines.

While drawing a polyline, set the type of control point for each segment either by clicking on the desired mode while drawing or by using the keyboard shortcuts (see "Creating or Editing Workspaces" on page 1835) to select the desired mode. A polyline can have different combinations of vertices. Use a polyline to create planar shapes with internal holes.

The corners of the polyline can be "smoothed" with the **Poly Smoothing** commands (see "Smoothing Objects" on page 1019). Markers can be added with the Attributes palette (see "Marker Attributes" on page 1102). The **Push/Pull** toggle mode is available in 3D views for instantly extruding the polyline after creation.



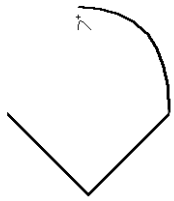
Mode	Description
Corner Vertex	Creates polyline segments with straight lines and angled vertices at the control points. The type of vertex created is a corner vertex.
Bézier Vertex	Creates polyline segments with curves pulled toward, but not touching the control points. The type of vertex created is a Bézier vertex.
Cubic Vertex	Creates polyline segments with curves that pass through the control points. The type of vertex created is a cubic vertex.
Tangent Arc	Creates polyline arc segments that are tangent to the previous segment (use tangent snapping to assist with drawing the tangent polyline; see "Snap to Tangent (2D Only)" on page 142). The type of vertex created is a radius vertex.
Point on Arc	Creates polyline arc segments that are drawn by clicking three points: the start point, a point the arc passes through, and the end point; useful for tracing existing arcs. The type of vertex created is a radius vertex.

Mode	Description
Arc Vertex Fillet	Creates polyline segments with curves that look like a fillet placed at the control points; click **Preferences** to set the radius of the fillet. The type of vertex created is an arc vertex.
Push/Pull (3D views only)	Instantly extrudes the polyline after creation; available in 3D views

To create a polyline:

1. Click the **Polyline** tool from the Basic palette.
2. Click to set the polyline's start point. In Point on Arc mode, click again to set a point that the arc will pass through.
3. Click to set the end of the segment and the beginning of the next.

While drawing a polyline, press the U key to switch the mode used to create the polyline



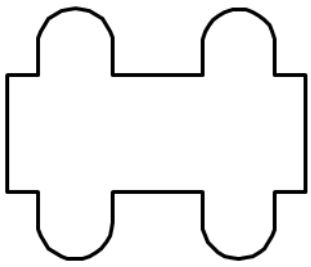
In Corner Vertex mode, press and hold instead of clicking to create a Bézier vertex instead of a Corner vertex.

4. Continue drawing segments in this manner until the polyline is complete.
5. Click the mouse at the start point to complete a closed polyline object (end point of the last segment is at the exact start point of the first segment), or double-click the mouse to complete an open polyline object (end point of the last segment is at a different location than the start point of the first segment).

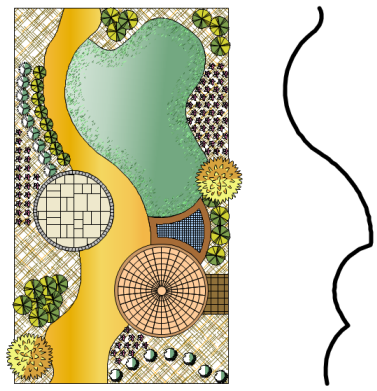
   Alternatively, after completing all but the final click, press the keyboard shortcut (K by default) to automatically close the polyline. The shortcut also closes path-based objects, such as hardscapes and spaces, that function similarly to the **Polyline** tool. See "Modifying Snapping and Mode Shortcuts" on page 1842 to change the shortcut.

   When drawing a polyline or path-based object, move the mouse in the direction of the next-to-last click and press the shortcut key before clicking; the software extrapolates the correct alignment and position for a 90° corner and places the final two clicks to complete the shape.

The resulting polyline object, whether open or closed, is a filled object. If desired, remove the fill through the Attributes palette to see objects behind the polyline.



Closed polyline created using Corner Vertex and Tangent Arc modes



Tracing of existing arcs created using Point on Arc mode

Edit polylines, changing their shape or their vertices, with the **Reshape** tool or the Object Info palette. The corners of the polyline can be "smoothed" with the **Poly Smoothing** commands (see "Smoothing Objects" on page 1019). Markers can be added with the Attributes palette (see "Marker Attributes" on page 1102).

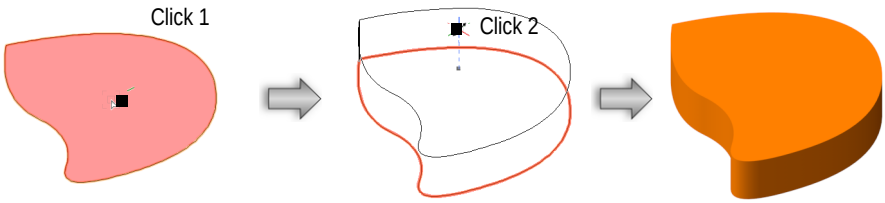
A closed polyline, when extruded with the **Model > Extrude** command, accepts a fill or texture and displays as a solid object. An open polyline, when extruded, does not render as a solid. To close a polyline before extruding it, select the polyline. In the Object Info palette, select the **Closed** option to close the polyline.

Creating Extruded Polylines

A planar polyline that has been created in a 3D view can be extruded immediately after creation.

To extrude a created polyline:

1. Enable the Push/Pull toggle mode in the Tool bar for the **Polyline** tool.
2. Create a polyline using any of the modes in the Tool bar.
3. With the polyline still selected, click, move the cursor, and then click to create the extrude.



A polyline can be extruded at any time with the **Push/Pull** tool. See "Direct Modeling with the Push/Pull Tool" on page 357.

The extrude height can be changed with the **Reshape** tool, or modified in the Object Info palette.

Reshaping Objects
Editing Vertex-Based Objects
Reshaping Extruded Objects and Solid Primitives
Closing and Opening Polygons and Polylines
Smoothing Objects
Push/Pull Mode of Planar Tools
Drawing Freehand Polylines
Simplifying Polygons and Polylines
Displaying and Reversing Object Direction

Drawing Freehand Polylines

Use the **Freehand** tool to draw a freehand polyline in a manner similar to drawing with a pen. Once the object is drawn, it can be reshaped.

Mode	Description
Freehand Edit	Reshapes the singularly selected polyline, polygon, rectangle, circle, or arc; for more information

Mode	Description
Preferences	Sets the smoothing level when drawing a curve. Increasing the degree of smoothing decreases the vertices, and therefore it is easier to reshape the curve. Decreasing the degree of smoothing increases the vertices which produces a more accurate representation of the curve. Select Off to draw the curve without using the smoothing feature.

To draw a freehand polyline:

1. Click the **Freehand** tool from the Basic palette.
2. Click **Preferences**.

   The Freehand Tool Preferences dialog box opens. Select the smoothing level when drawing a curve.
3. Click **OK**.
4. Click in the drawing to set the polyline start point. Move the mouse to create the desired freehand polyline shape.
5. Click again when the object is complete.

The number and placement of polyline vertices is determined by the object shape and specified degree of curve smoothing. For example, an object consisting of a series of arcs and curves created with a low degree of curve smoothing contains more vertices than a series of lines created with a high degree of curve smoothing.

Markers can be added with the Attributes palette (see “Marker Attributes” on page 1102).

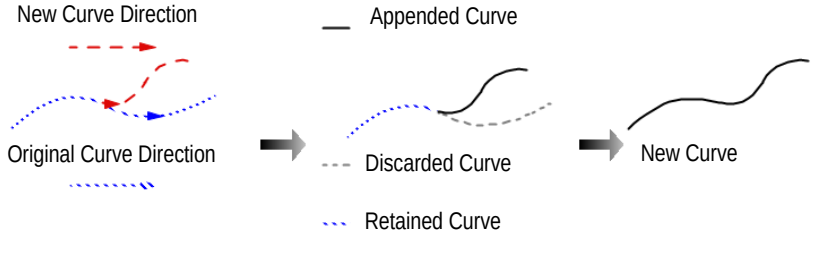
By default, the Freehand tool applies a fill of “None.” Change the fill type before sweeping a freehand polyline, if rendering of the sweep volume is desired.

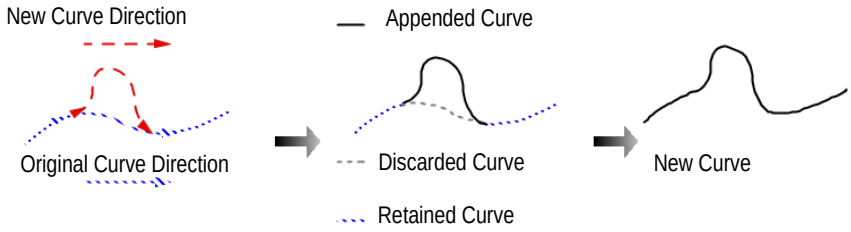
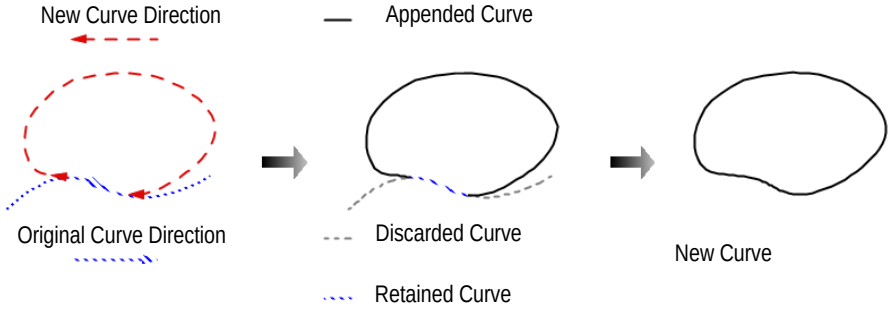
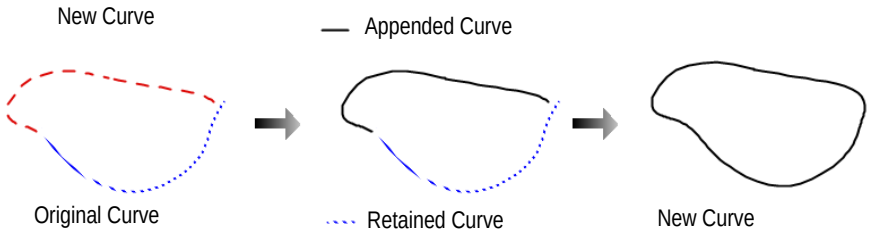
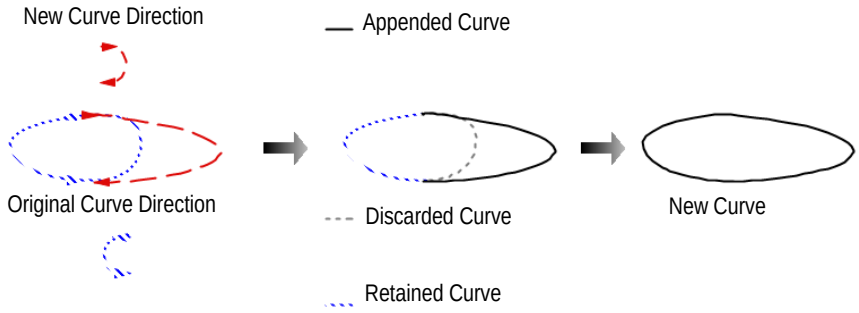
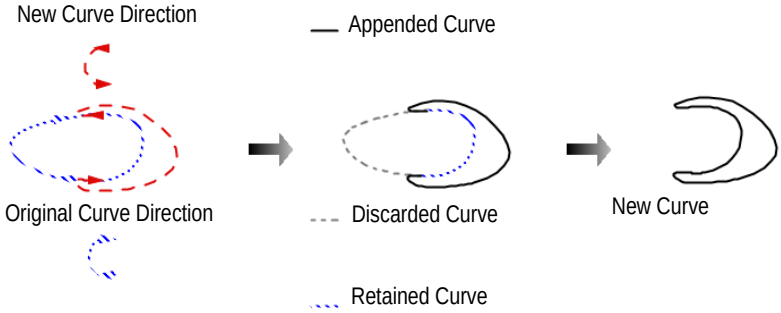
Reshaping Objects with the Freehand Tool

A polyline, polygon, rectangle, circle, or arc can be reshaped using the **Freehand Edit** mode of the **Freehand** tool. The direction of the curve drawn determines the new object shape. An edited object is converted to a polyline after editing is complete.

To reshape objects using the **Freehand** tool:

1. Select a valid object.
2. Click the **Freehand** tool from the Basic palette.
3. Click **Freehand Edit** mode from the Tool bar.
4. Click to draw the new curve, editing the valid object using the following curve direction guidelines:

Action	Guidelines
Change existing curve	

Action	Guidelines
Connect two points with new curve	
Create closure with new curve	
Add a new curve to an existing curve	
Extend existing curve	
Retain partial curve	

5. Click again to finish drawing the freehand polyline edit. The valid object is edited and converted to a polyline.

Polyline Tool
Reshaping Objects
Closing and Opening Polygons and Polylines
Displaying and Reversing Object Direction

Creating 2D Polygons

There are several ways to create 2D polygons. Single-line polygons, double-line polygons, and regular polygons can be created; a polygon always has square vertices and is composed of a series of straight line segments. A polygon that represents a curve will require more vertices (and more complexity) than a polyline.

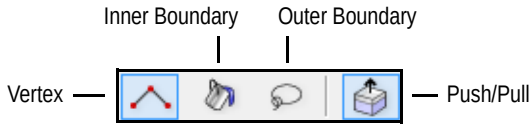
Polygons can be created automatically from existing geometry, which is especially useful for illustrating the elements of a hidden-line rendered viewport.

2D Polygon Tool
Double-Line Polygon Tool
Regular (Equal-sided) Polygon
Creating Extruded Polygons
Marker Attributes
Closing and Opening Polygons and Polylines
Simplifying Polygons and Polylines
Displaying and Reversing Object Direction

2D Polygon Tool

The **2D Polygon** tool creates open and closed polygons with single lines. Polygons can have as few as three vertices or as many as 32,767 vertices. The **2D Polygon** tool can also automatically create polygons by filling or outlining existing geometry, to easily annotate a drawing graphically by outlining, filling, or texturing (with an image or gradient fill) the new polygons.

Three modes are available. The **Push/Pull** toggle mode is available in 3D views for instantly extruding the polygon after creation.



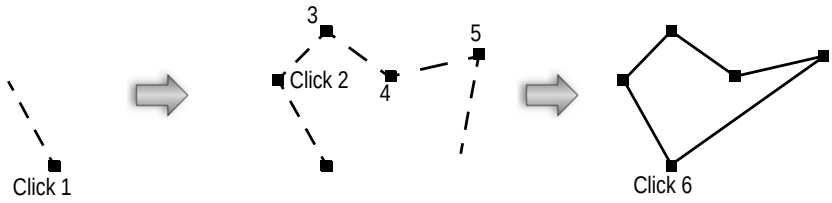
Mode	Description
Vertex	Creates a polygon by clicking to set each vertex
Inner Boundary	Creates a polygon out of existing geometry by clicking within the boundary of an object
Outer Boundary	Creates a polygon out of the outer boundary of existing geometry by defining geometry with a lasso marquee
Push/Pull (3D views only)	Instantly extrudes the polygon after creation; available in 3D views

Creating Single-line 2D Polygons

To create a single-line 2D polygon:

1. Click the **2D Polygon** tool from the Basic palette, and select **Vertex** from the Tool bar.

2. Click to set the polygon's start point (first vertex).
3. Click at each vertex.
4. Double-click at the final vertex to end an open polygon, or click at the starting vertex (a point cue displays) to end a closed polygon (the first and last vertex are automatically joined).



Alternatively, after completing all but the final click, press the keyboard shortcut (K by default) to automatically close the polygon. The shortcut also closes path-based objects that function similarly to the **Polygon** tool. See "Modifying Snapping and Mode Shortcuts" on page 1842 to change the shortcut.

When drawing a 2D polygon or path-based object, move the mouse in the direction of the next-to-last click and press the shortcut key before clicking; the software extrapolates the correct alignment and position for a 90° corner and places the final two clicks to complete the shape.

Creating a Polygon from an Inner Boundary

The **2D Polygon** tool can create a new polygon based on the inner boundary of existing geometry. Existing geometry refers to visible objects and solids in the active layer (within the active working plane) or the viewport cache of a hidden-line rendered viewport in Edit Annotation mode. The stacking order of the 2D objects does not apply; objects that are overlapped by other objects can still have their boundaries considered. If the object is a polyline (open or closed), a polygon with holes, or is curved, a polyline is created instead of a polygon.

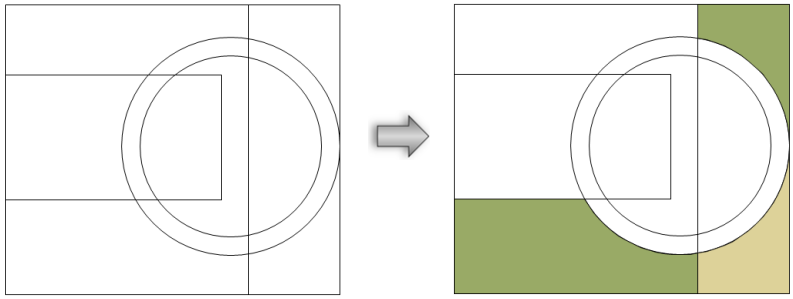
Polygons cannot be created from symbols. Convert the symbol to a group, and then ungroup.

To create a 2D polygon from the inner boundary of existing geometry:

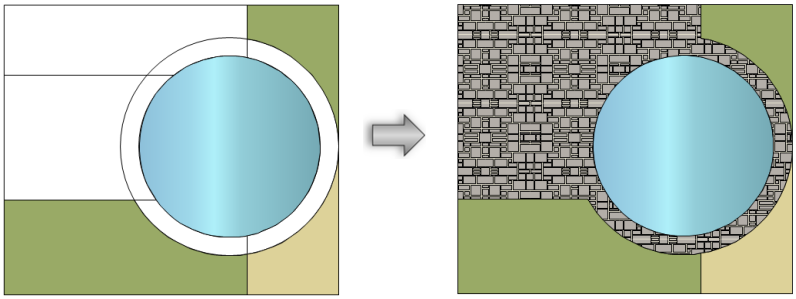
1. Click the **2D Polygon** tool from the Basic palette, and select **Inner Boundary** from the Tool bar.

   If desired, set the attributes in the Attributes palette (fill style, pen style, line and line endpoint style). The attributes of the polygon can also be specified after creation.
2. Click over one or more objects to create a polygon based on the inner boundary of the 2D objects and/or faces of solids in the active plane. Different methods of using the **2D Polygon** tool will achieve different results.

Modification	Method
Create a polygon	Click to place the polygon
Add to a polygon as it is created	Press the Shift key while clicking over existing adjacent geometry to add to the polygon
Apply the attributes of an existing object to the new polygon	Select the existing object and press the Option key (Mac) or the Alt key (Windows) while clicking to create the new polygon(s)



Create polygons from overlapping objects



Shift-click on a series of objects to create a single polygon

To speed up the polygon fill for complex images, zoom in on the area first.

The polygon overlays the existing geometry, and the original objects are unchanged

Another way to create planar objects from faces is with the Extract Surface mode of the **Extract** tool. See “Extracting Geometry” on page 345.

Creating a Polygon from an Outer Boundary

The **2D Polygon** tool can create a polygon based on the outer boundary of existing geometry. Existing geometry refers to visible objects in the active layer (within the active working plane) or the viewport cache of a hidden-line rendered viewport in Edit Annotation mode. The stacking order of 2D objects does not apply; objects that are overlapped by other objects can still have their boundaries considered. If the object is a polyline (open or closed), a polygon with holes, or is curved, a polyline is created instead of a polygon.

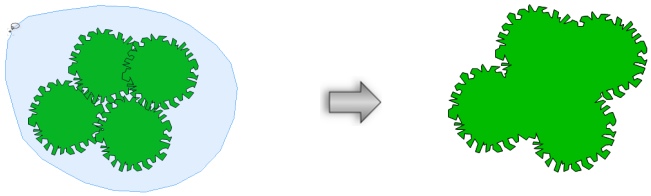
Polygons cannot be created from symbols. Convert the symbol to a group, and then ungroup.

To create a 2D polygon from the outer boundary of existing geometry:

1. Click the **2D Polygon** tool from the Basic palette, and select **Outer Boundary** from the Tool bar.

   If desired, set the attributes in the Attributes palette (fill style, pen style, line and line endpoint style). The attributes of the polygon can also be specified after creation.

2. Click in the drawing and drag to create the lasso marquee. A polygon is created based on the outer boundary of any 2D objects on the active plane completely enclosed within the marquee. Alternatively, press the Option key (Mac) or Alt key (Windows) while creating the marquee, and the polygon is based on the outer boundary of any co-planar 2D objects that are encountered by the marquee.



Creating Boundary Polygons in a Hidden Line Rendered Viewport

The creation of polygons based on the inner and outer boundaries of existing geometry is very useful for graphically annotating the elements of a sheet layer viewport that is rendered with hidden line rendering. (Sheet layer viewports are described in “Creating Sheet Layer Viewports” on page 1616.)

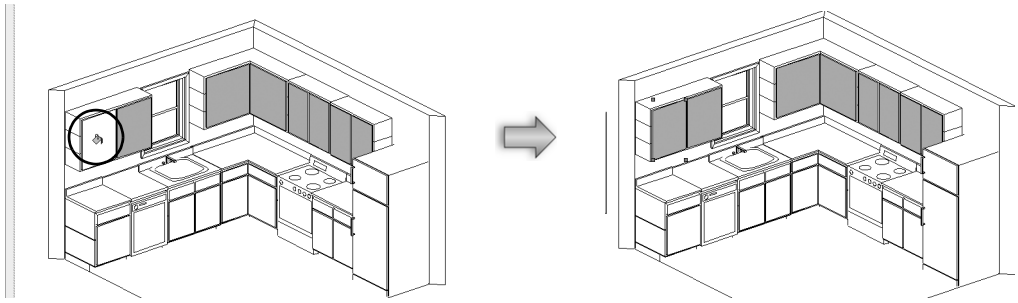
To create a 2D polygon from the inner/outer boundary of existing geometry in a viewport:

1. Create a sheet layer viewport from a design layer as described in “Creating a Sheet Layer Viewport from a Design Layer” on page 1616.
2. For the Rendering mode of the viewport, select **Hidden Line**. Update the viewport rendering by clicking **Update** from the Object Info palette.
3. Edit the viewport in annotation mode by selecting **Modify > Edit Viewport**.

   The Edit Viewport dialog box opens. Select **Annotations** and **Display Viewport Cache**.
4. Click **OK** to enter viewport annotation mode.
5. Click the **2D Polygon** tool from the Basic palette, and select **Inner Boundary** or **Outer Boundary** from the Tool bar.

   If desired, set the attributes in the Attributes palette to the desired fill settings (Fill Style, Pen Style, Line and Line Endpoint Style). The attributes of the polygon can also be specified after creation.

   Apply an image fill to the polygon to simulate a texture (see “Using Image Fills” on page 1119).
6. Because the viewport is rendered with hidden line and the viewport cache is used for annotations, any of the objects in the drawing can be used as the basis for new polygons. If in **Inner Boundary** mode, click the paint bucket cursor on the desired drawing objects. If in **Outer Boundary** mode, create a lasso marquee to include the desired objects. A 2D polygon is created based on the inner or outer boundary of the geometry.

   The **2D Polygon** tool works on 2D objects in the design layer. It does not apply to 2D objects or annotations that have been added to the sheet layer.
7. Click **Exit Viewport Annotation** at the top right corner of the drawing window to return to the sheet layer.

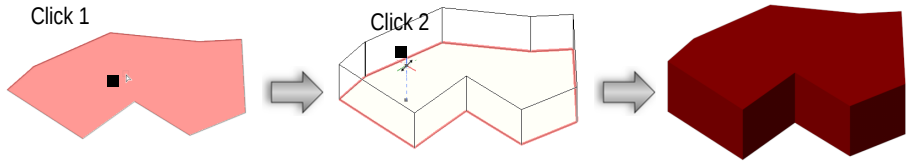
   An alert dialog box may ask if keeping the viewport cache is necessary. The viewport cache can be removed, if desired, as it is no longer necessary for creating the polygons.

Creating Extruded Polygons

A planar polygon that has been created in a 3D view can be extruded immediately after creation.

To extrude a created polygon:

1. Enable the Push/Pull toggle mode in the Tool bar for the **Polygon** or **Regular Polygon** tool.
2. Create a polygon using any of the modes in the Tool bar.
3. With the polygon still selected, click, move the cursor, and then click to create the extrude.



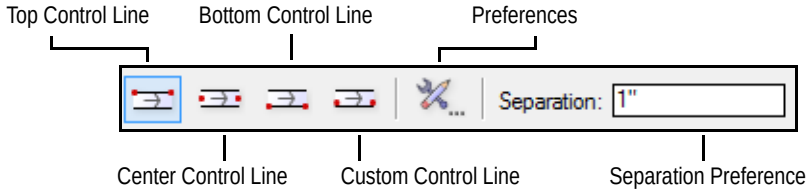
A polygon can be extruded at any time with the **Push/Pull** tool. See “Direct Modeling with the Push/Pull Tool” on page 357.

The extrude height can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), or modified in the Object Info palette.

Creating 2D Polygons
Regular (Equal-sided) Polygon
Push/Pull Mode of Planar Tools
Closing and Opening Polygons and Polylines
Simplifying Polygons and Polylines
Extracting Geometry

Double-Line Polygon Tool

Use the **Double-Line Polygon** tool to create a wide variety of open or closed polygons that have two parallel lines, a double-lined polygon with a width, or two parallel lines with a polygon fill. Double-line polygons can have as many as 32,767 vertices.



Mode	Description
Top Control Line	The cursor creates the right line
Center Control Line	Creates lines equidistant from the cursor
Bottom Control Line	The cursor creates the left line
Custom Control Line	Specify an offset value

To create a double-line polygon:

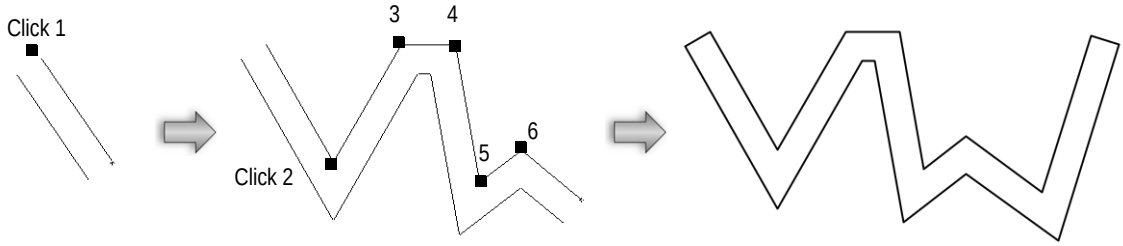
1. Click the **Double-Line Polygon** tool from the Basic palette.

2. Select the offset method.
3. Enter the distance between the double lines in the **Separation** field on the Tool bar.
4. Click the **Preferences** button to specify any other parameters.

Click to show/hide the parameters.

Parameter	Description
Separation	Enter the distance between the double lines; same as the **Separation** field on the Tool bar
Control Offset	For the Custom Control Line mode, enter the distance from the top/right line of the cursor
Options	
Create Lines	Creates a series of lines
Create Polygons	Creates a double-line polygon
Create Lines and Polygons	Creates parallel lines with a polygon between them; the polygon has a line weight of 0
Components	Click to define components between the double lines

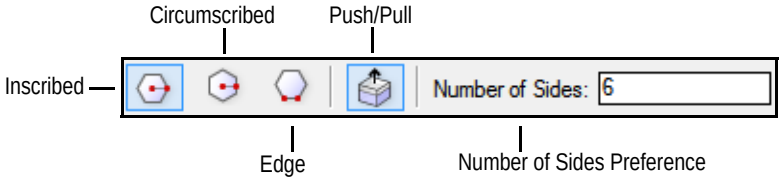
5. Click **OK**.
6. Click to set the polygon's start point (first vertex).
7. Click at each vertex.
8. Double-click to mark the polygon's end point (final vertex).



Applying Components Between Double Lines
2D Polygon Tool
Creating Double Lines
Regular (Equal-sided) Polygon
Displaying and Reversing Object Direction

Regular (Equal-sided) Polygon

Use the **Regular Polygon** tool to create single-line closed polygons in which all sides of the polygon are the same length.The **Push/Pull** toggle mode is available in 3D views for instantly extruding the polygon after creation.



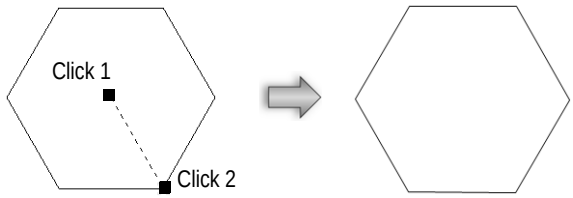
Mode	Description
Inscribed	Creates a polygon by drawing its radius to a corner
Circumscribed	Creates a polygon by drawing a radius to the mid-point of a side
Edge	Creates a polygon by drawing one of its sides
Push/Pull (3D views only)	After the polygon is created, click the selected polygon and drag to extrude it

Press and hold the Option (Mac) or Ctrl (Windows) key while drawing to toggle between circumscribed and inscribed modes.

Regular Polygon by Inscription

To create a regular polygon by inscription:

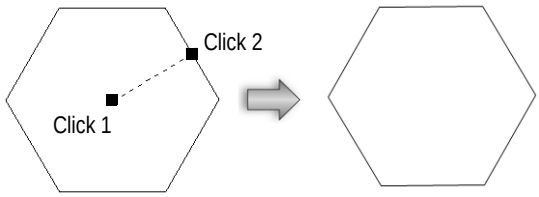
1. Click the **Regular Polygon** tool from the Basic palette and click **Inscribed** from the Tool bar.
2. Enter the **Number of Sides** for the polygon (a number from 3 to 4000).
3. Click to set the center of the polygon.
4. Click to set the corner radius.



Regular Polygon by Circumscription

To create a regular polygon by circumscription:

1. Click the **Regular Polygon** tool from the Basic palette and click **Circumscribed** from the Tool bar.
2. Enter the **Number of Sides** for the polygon (a number from 3 to 4000).
3. Click to set the center of the polygon.
4. Click to set the edge radius.

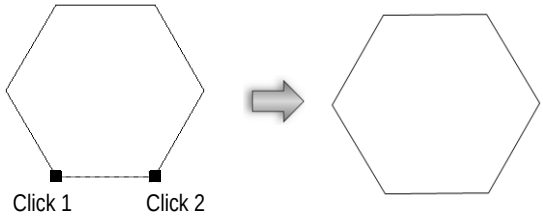


Regular Polygon by Edge

To create a regular polygon by edge:

1. Click the **Regular Polygon** tool from the Basic palette and click **Edge** from the Tool bar.
2. Enter the **Number of Sides** for the polygon (a number from 3 to 4000).

3. Click to set a corner of the polygon.
4. Click to set an adjacent corner of the polygon and define an edge.



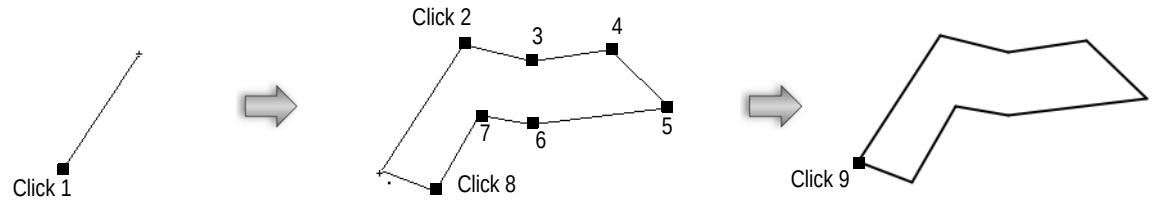
2D Polygon Tool
Double-Line Polygon Tool
Creating Extruded Polygons
Push/Pull Mode of Planar Tools
Displaying and Reversing Object Direction

Creating 3D Polygons

The **3D Polygon** tool creates polygons that have a location in 3D space, but no height. The polygon can be created through any 3D point, not necessarily constrained to the working plane.

To create a planar 3D polygon:

1. Click the **3D Polygon** tool from the 3D Modeling tool set.
2. Click to set the polygon's start point (first vertex).
3. Click at each vertex.
4. Double-click at the final vertex to end an open polygon; click at the starting vertex (a point cue displays) to end a closed polygon (the first and last vertex are automatically joined).



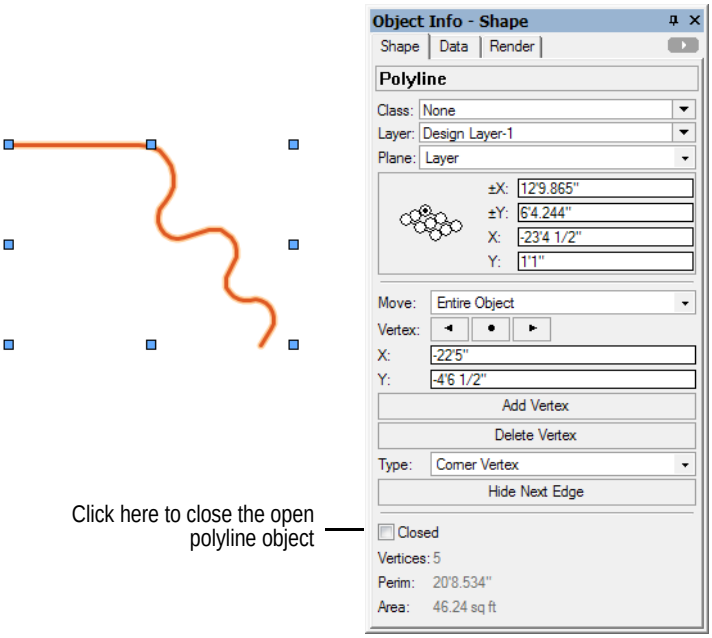
Click here for a video tip about this topic (Internet access required).

Creating Extruded Polygons
Closing and Opening Polygons and Polylines
Displaying and Reversing Object Direction

Closing and Opening Polygons and Polylines

When created, 2D and 3D polygon and polyline objects can be either open or closed. For polylines, you can also use the **Hide Next Edge** button on the Object Info palette, or use the **Hide or Show Edges** mode of the **Reshape** tool to hide specific edges of the object after creation, which opens a closed object.

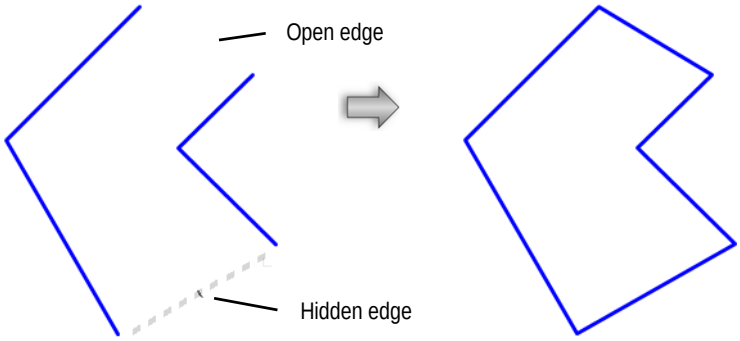
Use the **Closed** setting on the Object Info palette to change the current state of the object.



The **Closed** setting is available for objects created with the following tools:

- Freehand
- Polyline
- 2D Polygon
- Triangle
- Double-Line Polygon
- Regular Polygon
- 3D Polygon

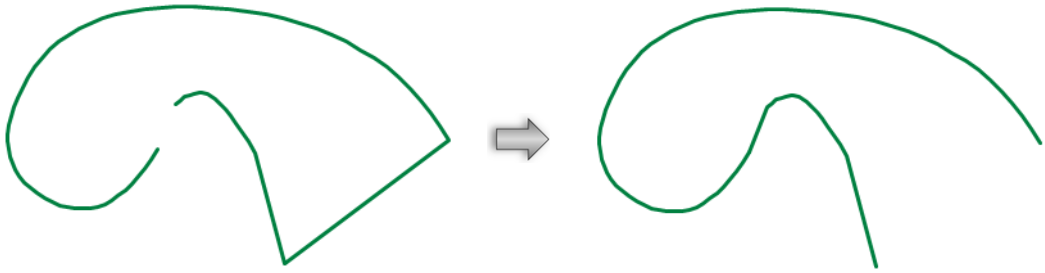
If a polygon or polyline is open, select the **Closed** setting to close it; any edges that were hidden are also closed.



When the Closed setting is selected, the open and hidden edges of the polygon are closed

If a polygon or polyline is closed, deselect the **Closed** setting to open it. The last segment of the object to be drawn is removed (for polygons) or hidden (for polylines).

For polylines, the last segment of the object must be between vertices created with **Corner Vertex**, **Tangent Arc**, or **Point on Arc** modes; otherwise, the polyline cannot be opened by deselecting the **Closed** setting. To open a segment other than the last segment, use the **Hide or Show Edges** mode of the **Reshape** tool.



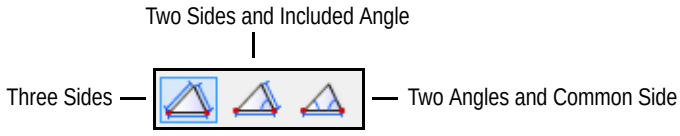
When the **Closed** setting is deselected, the last segment to be drawn is opened

To hide a different segment, use the **Reshape** tool in **Hide or Show Edges** mode

Reshaping Objects

Creating Triangles

The **Triangle** tool has three modes. The steps for drawing the triangle are the same for each mode; the only difference is the fields that display in the dialog box that opens.



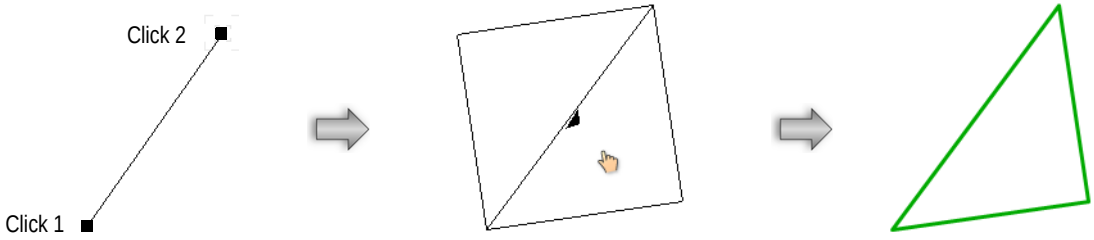
To create a triangle:

1. Click the **Triangle** tool from the Basic palette and select the desired mode.
2. Click to set the first and second points of the first side.

   The Triangle Settings dialog box opens; the fields available depend on the mode that was selected.
3. The length of the first side of the triangle displays. Enter the remaining fields as required for the selected mode and click **OK.** The following restrictions apply.

Mode	Restriction
Three Sides	The sum of the lengths of any two sides of the triangle must be greater than the remaining side
Two Sides and Included Angle	The angle must be less than 180 degrees
Two Angles and Common Side	The sum of the two angles must be less than 180 degrees

4. Two possible triangles display; click the triangle to keep.



Click two points to define the first side of the triangle

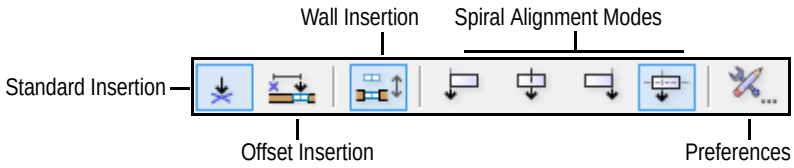
After you enter the required side lengths and angles, two possible triangles display; click the triangle to keep

Click here for a video tip on this topic (Internet access required).

Closing and Opening Polygons and Polylines

Creating Spirals

The **Spiral** tool draws an Archimedes spiral. The number of turns, distance per turn, start radius and thickness can be specified, as well as the number of points used to define the curve. Use the alignment modes on the Tool bar to temporarily override the insertion point. These modes change the alignment of the insertion point along the X axis of the bounding box surrounding the spiral.



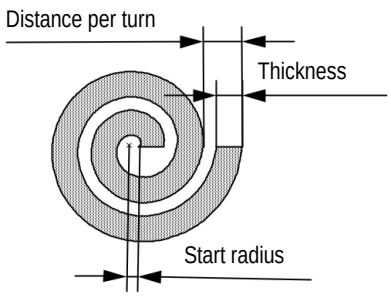
Spiral Alignment Mode	Description
Align Object Left	Moves the insertion point to the left edge of the spiral's bounding box, along the original X axis
Align Object Center	Moves the insertion point to the center of the spiral's bounding box, along the original X axis
Align Object Right	Moves the insertion point to the right edge of the spiral's bounding box, along the original X axis
Align Object Origin	Leaves the insertion point at the actual or original position

For information on using the Offset Insertion and Wall Insertion modes, see "Offset Symbol Insertion Mode" on page 245 and "Wall Insertion Mode" on page 246.

To draw a spiral:

1. Click the **Spiral** tool from the Basic palette.
2. Select the insertion type and alignment from the Tool bar.
3. Click to define the center of the spiral.

   If this is the first time a spiral is placed in this session, the Spiral Properties dialog box opens. These parameters apply to subsequently created spirals; they can be changed later by accessing them from the Object Info palette.
4. Specify the spiral properties.



Click to show/hide the parameters.

Parameter	Description
Distance per Turn	Enter the distance between the outer edges of each turn in the spiral
Number of Turns	Specify the number of turns which determine the total sweep angle of the spiral; one turn equals 360 degrees
Start Radius	Enter the distance from the center to the beginning of the spiral
Increment (deg)	Specify the number of points used to define the curve; the higher the increment, the fewer the number of points (for example, an increment of five degrees means 360/5=72 points per turn)
Thickness	Specify the thickness value of the area between the outer and inner edge of the turn

5. Click **OK**.

   A spiral with the specified parameters is placed on the drawing.

   To create a 3D spiral, see "Creating Helix-Spirals" on page 355.

   Click here for a video tip on this topic (Internet access required).

Creating Loci

A locus is a reference point that is used to draw and measure objects. Loci do not print.

Turn on **Snap to Objects** from the Snapping palette to snap to loci (see "Object Snapping" on page 134).

2D Locus Tool

The **2D Locus** tool places a 2D locus in a drawing. Because they are merely movable reference points, loci cannot be reshaped or resized.

To place a 2D locus:

1. Click the **2D Locus** tool from the Basic palette.
2. Click to place the locus.

   The locus properties can be edited from the Object Info palette.

3D Locus Tool

The **3D Locus** tool places a snappable 3D locus or reference point onto the drawing. Like 2D loci, they are reference points. They can be moved, but they cannot be reshaped and they do not print.

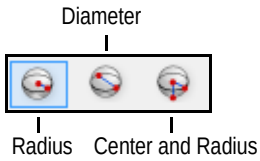
To place a 3D locus:

1. Click the **3D Locus** tool from the 3D Modeling tool set.
2. Click to place the locus.

   The locus is placed on the working plane unless it is snapped to an object. The locus properties can be edited from the Object Info palette.

Creating Spheres

The **Sphere** tool creates spheres using one of three modes.



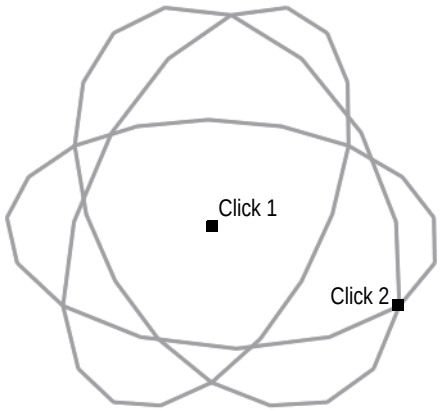
Mode	Description
Radius	Defines the base of the sphere by radius
Diameter	Defines the base of the sphere by diameter
Center and Radius	Defines the base of the sphere by center (according to the height above the working plane) and radius

The sphere radius can be changed with the **Reshape** tool (see "Reshaping Extruded Objects and Solid Primitives" on page 1052), the radius can be modified in the Object Info palette, and the resolution can be adjusted through the **3D Conversion Resolution** setting (see "3D Preferences" on page 54).

Sphere by Radius

To create a sphere by radius:

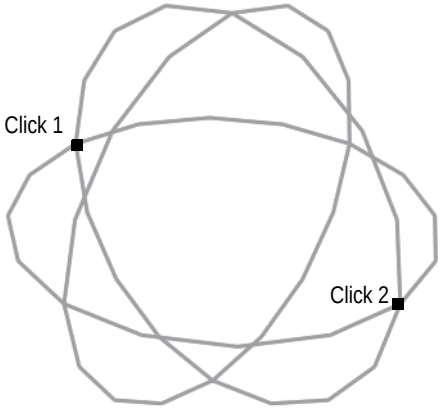
1. Click the **Sphere** tool from the 3D Modeling tool set, and select the **Radius** mode.
2. Click to set the center of sphere base.
3. Move the mouse to the desired radius and click to set the radius of the sphere base. The radius can also be set in the Data bar.



Sphere by Diameter

To create a sphere by diameter:

1. Click the **Sphere** tool from the 3D Modeling tool set, and select the **Diameter** mode.
2. Click to set the first point on the sphere base diameter.
3. Move the mouse to the desired diameter and click to set the diameter of the sphere base.



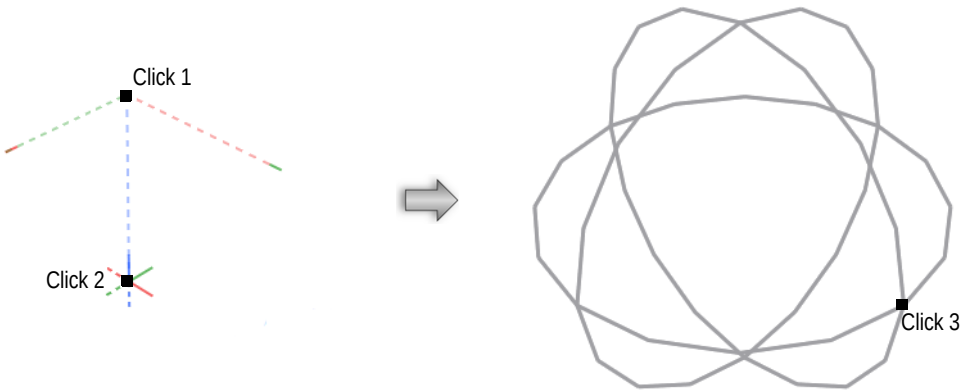
Sphere by Center and Radius

To create a sphere by center and radius:

1. Click the **Sphere** tool from the 3D Modeling tool set, and select the **Center and Radius** mode.
2. Click to set the center of the sphere base on the working plane.
3. Move the mouse and click to set the height of the sphere above the working plane.
4. Move the mouse to the desired radius and click to set the radius of the sphere base. The radius can also be set in the Data bar.

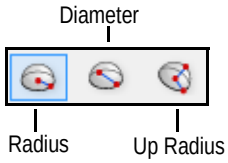
The center and height, relative to the working plane, of the sphere is automatically established if the first click is on a snap point—for example a 3D locus or vertex on another 3D object.

When in a non-isometric view, the sphere is created centered on the working plane.



Creating Hemispheres

The **Hemisphere** tool creates hemispheres using one of three modes.



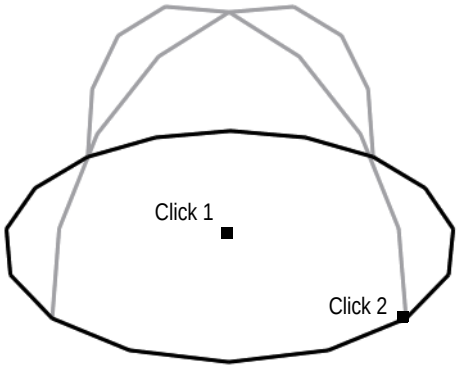
Mode	Description
Radius	Defines the base of the hemisphere by radius
Diameter	Defines the base of the hemisphere by diameter
Up Radius	Defines the base of the hemisphere by center and rotation

The hemisphere radius can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), the radius can be modified in the Object Info palette, and the resolution can be adjusted through the **3D Conversion Resolution** setting (see “3D Preferences” on page 54).

Hemisphere by Radius

To create a hemisphere by radius:

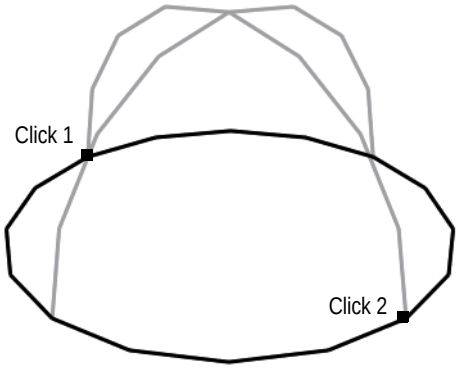
1. Click the **Hemisphere** tool from the 3D Modeling tool set, and select the **Radius** mode.
2. Click to set the center of the hemisphere base.
3. Move the mouse to the desired radius and click to set the radius of the hemisphere base. The radius can also be set in the Data bar.



Hemisphere by Diameter

To create a hemisphere by diameter:

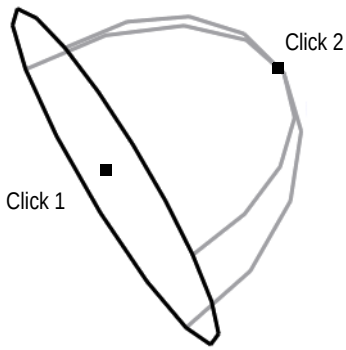
1. Click the **Hemisphere** tool from the 3D Modeling tool set, and select the **Diameter** mode.
2. Click to set the first point on the hemisphere base diameter.
3. Move the mouse to the desired diameter and click to set the diameter of the hemisphere base.



Hemisphere by Up Radius

 To create a hemisphere by up radius:

1. Click the **Hemisphere** tool from the 3D Modeling tool set, and select the **Up Radius** mode.
2. Click to set the center of the hemisphere base.
3. Move the mouse to the desired radius and click to set the rotation and top of the hemisphere.



Creating Cones

The **Cone** tool creates cones using one of two modes.

Mode	Description
Radius and Height	Defines the base of the cone by radius and tip of the cone by height
Radius and Tip	Defines the base of the cone by radius and, if desired, snaps the tip of the cone to a point

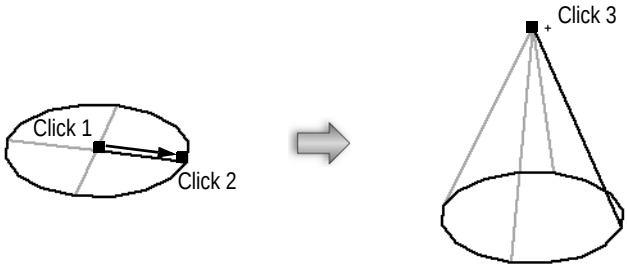
When in a non-isometric view, the height of the cone is set in the Set Extrusion Height dialog box. In an isometric view, the cone height can be set by dragging with the mouse or entering the height in the Data bar. For more information on the different views, see “Using Standard Views” on page 1141.

The cone height can be changed with the **Reshape** tool (see “Reshaping Extruded Objects and Solid Primitives” on page 1052), the radius can be modified in the Object Info palette, and the resolution can be adjusted through the **3D Conversion Resolution** setting (see “3D Preferences” on page 54).

Cone by Radius and Height

To create a cone using radius and height:

1. In a 3D isometric view, click the **Cone** tool from the 3D Modeling tool set, and select the **Radius and Height** mode.
2. Click to set the center of the cone base.
3. Move the mouse to the desired radius and click to set the radius of the cone base. The radius can also be set in the Data bar.
4. Move the mouse and click to set the cone height.



Cone by Radius and Tip

To create a cone using radius and tip:

1. Ensure that Snap to Object and Smart Points snapping is enabled (see “Setting Snapping Parameters” on page 131).
2. In a 3D isometric view, click the **Cone** tool from the 3D Modeling tool set and select the **Radius and Tip** mode.
3. Click to set center of the cone.
4. Move the mouse to the desired radius and click to set the radius of the cone base. The radius can also be set in the Data bar.
5. Using snapping, snap the top of the cone to a snap point on another 3D object. A preview object displays. Click to create the cone.

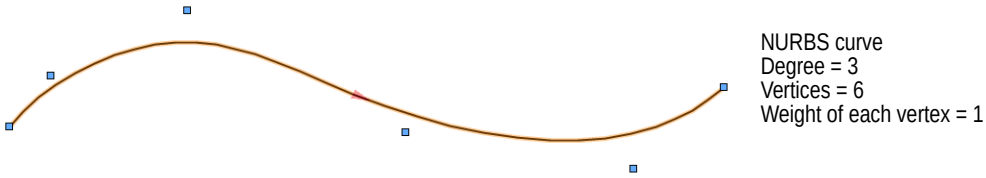
Click here for a video tip on this topic (Internet access required).

Advanced Object Creation

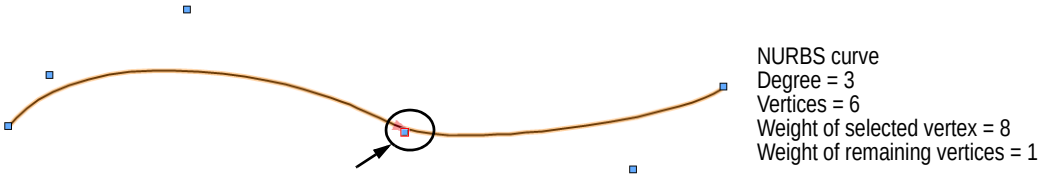
NURBS Curves and Surfaces

Non-Uniform Rational B-Spline (NURBS) is a mathematical formulation that represents the geometry of curves, circles, arcs, and surfaces in 3D space. Free-form curves and surfaces can be created and edited with a high level of both flexibility and precision.

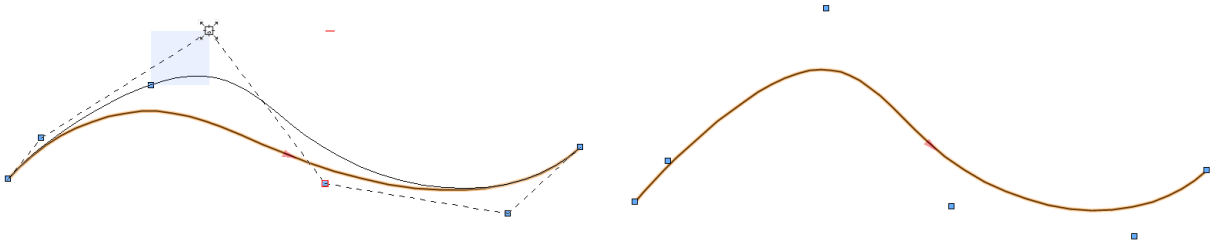
A NURBS curve generally consists of a degree value and weighted control points, or vertices. The curve passes between the vertex points; the degree determines how many points affect the curve. The direction indicates the starting and ending points of the curve as it was drawn, which can affect the outcome of certain operations.



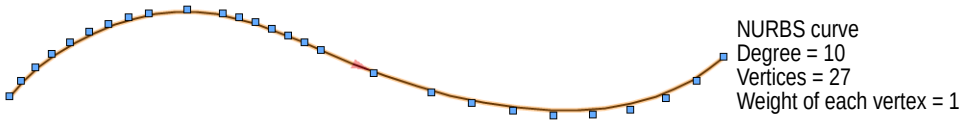
Increasing the relative weight of one of the vertices means the vertex has more of an influence on the curve and "pulls" the curve towards that vertex.



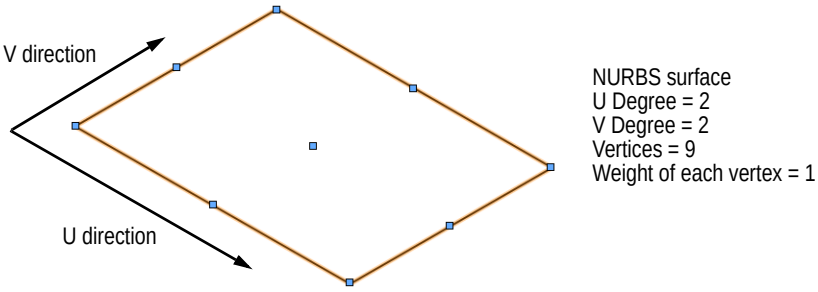
The **Reshape** tool can be used to move a vertex or several vertices, changing the shape of the curve (see "Reshaping NURBS Curves" on page 1054).



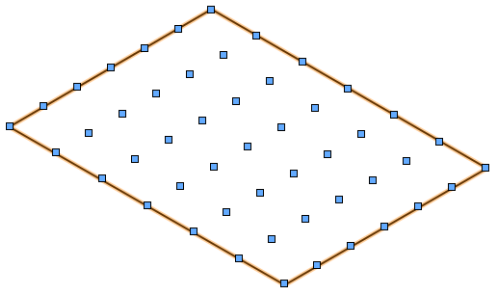
Increasing the degree of a NURBS curve proportionally increases the number of vertices, allowing for more flexibility in drawing the curve, but also more complexity due to the numerous weights affecting the curve.



The same principles that apply to a NURBS curve apply to a surface. A NURBS surface is a grid, or mesh, of weighted control points in the U and V directions.

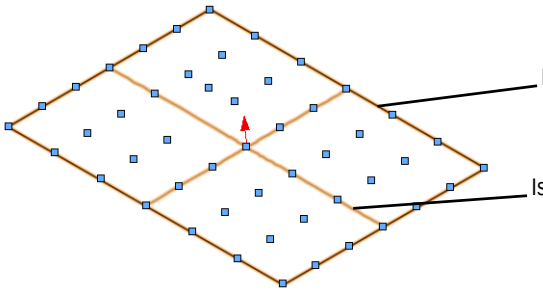


Increasing the degree of a NURBS surface in the U and/or V direction increases the number of vertices, adding flexibility as well as complexity.

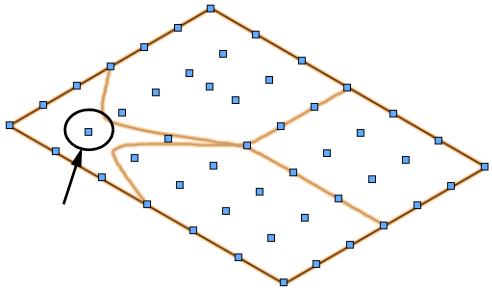


NURBS surface
U Degree = 6
V Degree = 6
Vertices = 49
Weight of all each vertex = 1

A NURBS surface also has a direction, or normal, which affects the outcome of certain operations, such as fillet surface creation. Iso-parametric curves indicate the U and V direction, and edge curves are drawn along the edges, helping to visualize the NURBS surface.

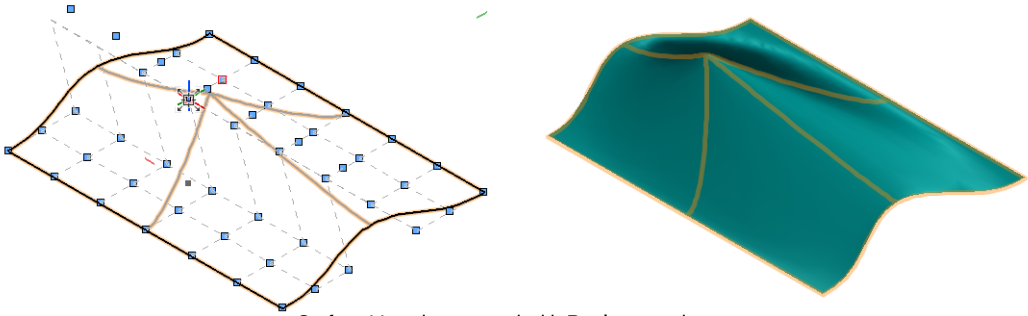


Each vertex on the surface can have a weight which "pulls" the surface towards the weighted vertices.



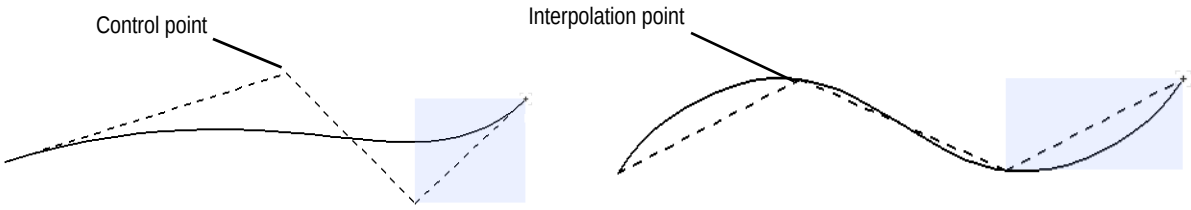
NURBS surface
U Degree = 6
V Degree = 6
Vertices = 49
Weight of selected vertex = 100
Weight of remaining vertices = 1

The **Reshape** tool can move a single vertex or a row of vertices, deforming the surface (see "Reshaping NURBS Surfaces" on page 1055).



Surface V vertices moved with **Reshape** tool

Both NURBS curves and surfaces can be defined by control points or by interpolation points. The curve or surface passes *between* control points or *through* interpolation points.



Interpolated curves and surfaces may be easier to modify. See “Creating NURBS Curves” on page 328 and “Interpolated NURBS Surfaces” on page 329.

Complex, free-form shapes can be created with NURBS curves and NURBS surfaces. The shapes can then be combined, clipped, added to, trimmed, extended, analyzed, and otherwise modified as described in these sections.

Click here for a video tip on this topic (Internet connection required).

3D Power Pack Cursors
Selecting the Edges and Faces of a Solid
Displaying Surface Normals
Displaying and Reversing Object Direction
Editing a Fillet/Chamfer or Shell Object
Surface Geometry Requirements
Converting to NURBS
NURBS Surface Properties

3D Power Pack Cursors

Special cursors display during certain operations to help determine the current mouse function; they do not display when an active selection is about to occur.

Cursor/Operation	Where Used
Edge Selection 	**Extract** tool’s **Point** and **Curve** modes, **Chamfer Edge** tool, and the **Fillet Edge** tool
Face Selection 	**Extract** tool’s **Surface** modes, **Shell Solid** tool, **Push/Pull** tool, as well as the **Chamfer Edge** and **Fillet Edge** tools with the Select Faces option turned on. Press the Alt key (Windows) or Option key (Mac) to select the back faces of solid objects.
Curve Selection 	**Loft Surface** tool, **Project** tool, and **Push/Pull** tool
Surface Selection 	**Project** tool

NURBS Curves and Surfaces
Selecting the Edges and Faces of a Solid
Displaying Surface Normals
Displaying and Reversing Object Direction
Editing a Fillet/Chamfer or Shell Object
Surface Geometry Requirements

Converting to NURBS

Selecting the Edges and Faces of a Solid

Several 3D Power Pack tools require the selection of edges or faces of a solid. For example, the **Extract** tool's Extract Surface mode, the **Push/Pull** tool, and the **Shell Solid** tool require face selection. The **Chamfer Edge** and **Fillet Edge** tools can require both face and edge selection.

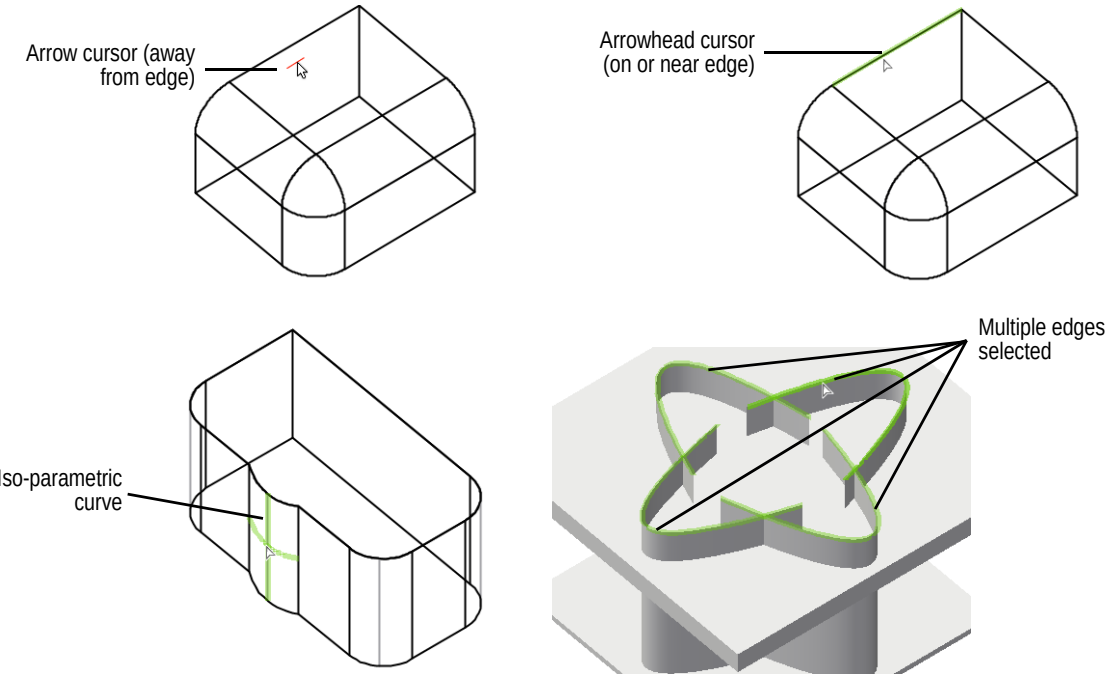
The color and opacity used for pre-selection highlighting is set by the Object Highlighting - Tool Highlighting preference in the interactive settings. See "Selection and Pre-selection Indicators" on page 114 and "Configuring Interactive Display" on page 116.

Selecting Edges

The cursor changes to an arrowhead when on or near an edge, and the edge is highlighted for selection. If more than one edge is near the cursor, the nearest edge is highlighted. Click the edge to select it.

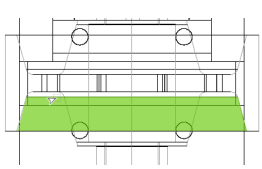
The cursor also changes to an arrowhead when on or near an iso-parametric curve for the **Extract** tool in Extract Iso-parametric Curves mode.

Action	Description
Select more than one edge	Press the Shift key and select the edges
Deselect a selected edge	Click on the edge again with the Shift key pressed
Deselect edges that have been selected	Click on an empty area
Deselect the last selection	Press the Backspace key, or double-click on the edge

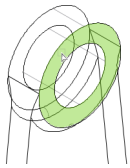


Selecting Faces

When the cursor is over a face, the face geometry is highlighted, making it very easy to determine which face will be selected. Click on a face to select it.



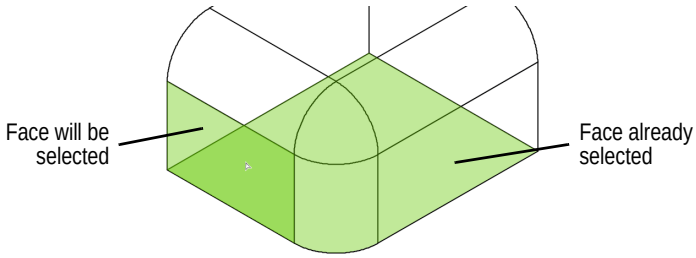
2D face selection



Wireframe 3D face selection

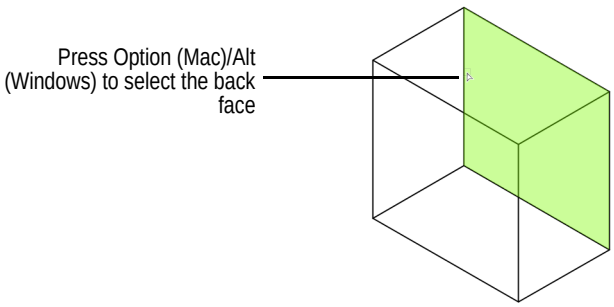
Rendered 3D face selection

Multiple faces can be selected while pressing the Shift key, much like the edge selection described earlier. Deselecting faces is also similar to deselecting edges.

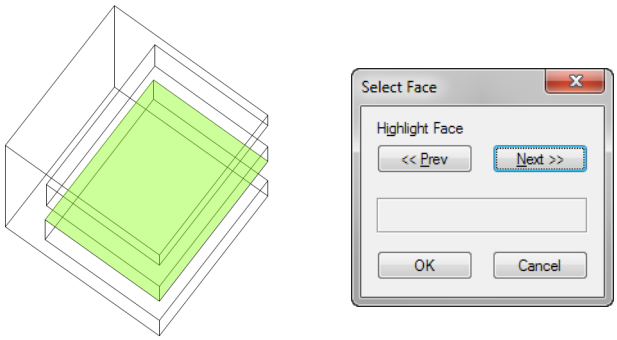


Face selection can also be an edge-based selection. When the cursor is on or near an edge or near an iso-parametric curve, you can select the face on either side of the edge depending on the cursor position. Face highlighting helps determine which face will be selected.

Normally, only the front surfaces of a solid object can be selected directly. To select the back face of a solid, press the Option key (Mac) or the Alt key (Windows) during face selection.



If the cursor is positioned over more than one back face and the desired face to select is difficult to determine, the Select Face dialog box opens to specify the face. Click **Next** or **Prev** until the desired face is highlighted. Click **OK** to select the face.



NURBS Curves and Surfaces
3D Power Pack Cursors
Displaying Surface Normals

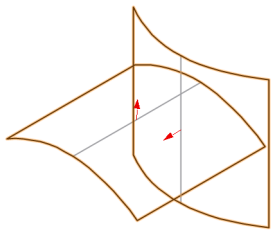
Displaying and Reversing Object Direction
Editing a Fillet/Chamfer or Shell Object
Surface Geometry Requirements
Configuring Interactive Display
Converting to NURBS

Displaying Surface Normals

The surface normal of NURBS surfaces can be displayed in order to clarify the surface direction and facilitate the creation of fillet surfaces (see "Creating a Fillet Surface" on page 338), shell solids (see "Shell from a NURBS Surface" on page 371), and the sectioning of solids (see "Section Solids" on page 1086).

To display the surface normal of a NURBS surface:

1. Select one or more NURBS surfaces.
2. In the Object Info palette, select **Show Normal**. The NURBS surface normal displays as a red arrow.



If desired, click **Reverse Normal** when a single NURBS surface is selected to reverse the direction of the surface normal. The arrow changes direction accordingly to indicate the new direction.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

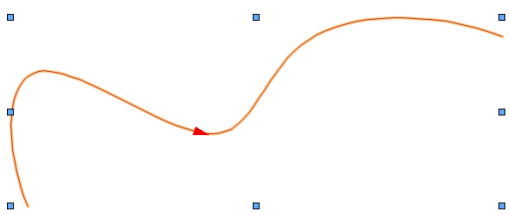
NURBS Curves and Surfaces
3D Power Pack Cursors
Selecting the Edges and Faces of a Solid
Displaying and Reversing Object Direction
Editing a Fillet/Chamfer or Shell Object
Surface Geometry Requirements
Converting to NURBS

Displaying and Reversing Object Direction

The direction of lines, polylines, 2D/3D polygons, and NURBS curves can be displayed and reversed, if desired, to help perform 3D operations and make end marker placement predictable.

To display the direction of an object:

1. Select one or more objects.
2. In the Object Info palette, select **Show Direction**. The object's direction displays as a red arrow.



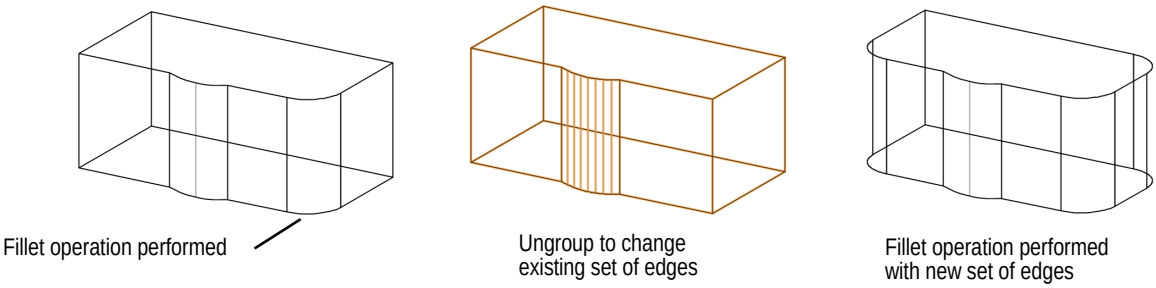
If desired, click **Reverse Direction** with one or more objects selected to reverse the direction of the object(s). The arrow indicates the new direction.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

NURBS Curves and Surfaces

Editing a Fillet/Chamfer or Shell Object

A fillet, chamfer, or shell object follows certain editing conventions.

The set of edges or faces used to create the fillet/chamfer or shell cannot be changed once the operation is performed; edges or faces cannot be deleted from or added to the existing set. To add or delete edges or faces, first ungroup the object, and then perform the operation again.



The properties that can be changed from the Object Info palette include the thickness of a shell, direction of shelling (inside/outside), the radius of a fillet or setback distance of a chamfer. In the case of variable radius fillets, the percentage of length and radius value at each point can be edited. However, only the parameters of the topmost object can be changed.

For example, if a shell is created, and then some of the shell edges are filleted, only the fillet parameters can be changed directly from the Object Info palette. To change the shell thickness, the fillet object must first be ungrouped. To change the original extrusion, both the fillet and the shell must be ungrouped. Once changes have been made, reapply the shell and fillet.

The **Modify > Edit Group** command cannot be used for these objects.

3D Power Pack Cursors
Selecting the Edges and Faces of a Solid
Surface Geometry Requirements

Surface Geometry Requirements

Some surface geometry can contain degenerate patches, singularities, or self-intersections. These types of surfaces could potentially be produced with the **Reshape** tool, **Loft Surface** tool, or **Create Surface from Curves** command, and can produce undesirable results in the finished model. Surface operations, such as trim and stitch surfaces, split, and other operations, like creating contours and solid operations, may not be able to manipulate these types of surfaces.

Decomposing such surfaces may help by separating the surfaces into NURBS surfaces without discontinuities.
See “Decomposing Objects and Surfaces” on page 1020.

NURBS Curves and Surfaces

Converting to NURBS

The **Convert to NURBS** command converts planar objects and 3D polygons into NURBS curves, and converts the faces of solids to NURBS surfaces. This is a quick way of creating NURBS curves from circles and arcs, or NURBS surfaces from an extrusion, sweep, or other solids.

To convert to NURBS curves or surfaces:

1. Select the object or solid to convert to NURBS surfaces.
2. Select **Modify > Convert > Convert to NURBS**.

   The selection is converted to NURBS surface(s) or a NURBS curve, as reflected in the Object Info palette.

If the solid consisted of several faces, the conversion results in a group of NURBS surfaces. Select **Modify > Ungroup** to access the individual NURBS surfaces that make up the solid.

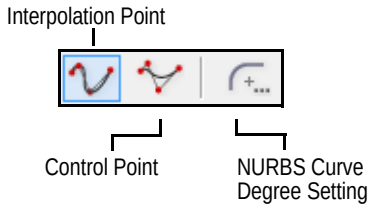
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

NURBS Curves and Surfaces
3D Power Pack Cursors
Selecting the Edges and Faces of a Solid
Displaying Surface Normals

Creating NURBS Curves

NURBS (Non-Uniform Rational B-Splines) are used to create curves in 3D space. They can also be used as defining objects for extrusions along a path.

NURBS curves can be created using one of two modes. Specify the NURBS curve degree prior to selecting a mode.



NURBS curves can also be created from planar objects by converting them to NURBS; see “Converting to NURBS” on page 327.

Setting the NURBS Curve Degree

The curve degree affects the number of vertices created; the higher the degree value, the greater the number of vertices.

To specify the NURBS curve degree:

1. Click the **NURBS Curve** tool from the 3D Modeling tool set, and click NURBS Curve Degree Setting from the Tool bar.

   Alternatively, double-click the **NURBS Curve** tool to open the Curve Degree dialog box.
2. The Curve Degree dialog box opens. Specify the curve degree for the **NURBS Curve** tool to use. Click **OK**.

   Set the degree value to 1 to create a NURBS curve with linear segments (similar to a 3D polygon). A linear NURBS curve can be used to measure an object from point to point in a 3D view.

NURBS Curve by Interpolation Point

To create a NURBS curve by interpolation point:

1. Click the **NURBS Curve** tool from the 3D Modeling tool set, and select the **Interpolation Point** mode.
2. Click to set the first point on the curve.
3. Click to set the point through which the curve passes.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Continue clicking to add more interpolation points, extending the curve.

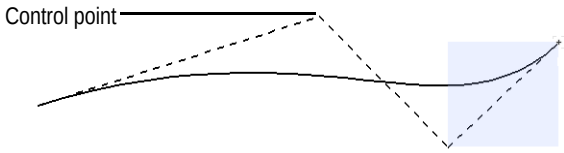
4. Double-click to set the end point of the curve.

NURBS Curve by Control Point

To create a NURBS curve by control point:

1. Click the **NURBS Curve** tool from the 3D Modeling tool set, and select the **Control Point** mode.
2. Click to set the first point on the curve.
3. Click to set the point that the curve pulls toward but does not touch.

   Continue clicking to add more control points, extending the curve.



4. Double-click to set the end point of the curve.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Reshaping NURBS Curves
NURBS Curves and Surfaces

NURBS Surfaces

Interpolated NURBS Surfaces

As described in “Creating NURBS Curves” on page 328, NURBS curves can be created by interpolation points or control points. Similarly, a NURBS surface can be created by interpolation points. An interpolated surface is a NURBS surface that passes through a two-dimensional array of 3D interpolation points. Regular NURBS surfaces are defined by their control points, which may not lie on the surface and can be difficult to use for reshaping the surface (it is difficult to know how much the control point should be moved in order to reshape the surface by a specific distance). Because interpolation points lie on the surface, it is much easier to modify these points with the **Reshape** tool or the Object Info palette, and have the surface pass through the points.

An interpolated surface can be created, or an existing untrimmed NURBS surface can be converted to an interpolated surface, for easier reshaping.

> When an interpolated surface is used in another operation, such as a Boolean operation or trimming, the surface becomes a control point surface.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

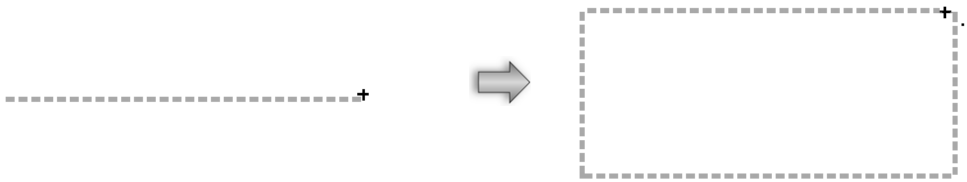
Creating an Interpolated Surface
Converting to an Interpolated Surface
Creating a Loft Surface
NURBS Curves and Surfaces

Creating an Interpolated Surface

A new interpolated NURBS surface can be created with the **Create Interpolated Surface** command.

To create an interpolated surface:

1. Select **Model > 3D Power Pack > Create Interpolated Surface**.
2. Click to define the start of the interpolation point row, and then click again to define the end of the row. Click a third time to define the column of interpolation points.



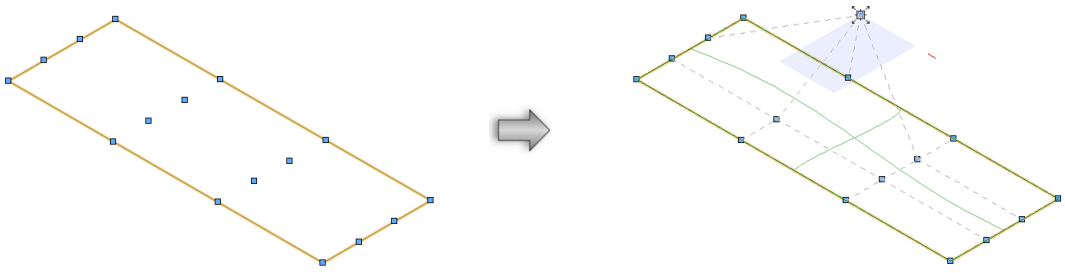
The Create Interpolated Surface dialog box opens.

3. Specify the number of interpolation points and degree of flexibility to create for both the U and V direction.

Click to show/hide the parameters.

Parameter	Description
U Direction	
Number of Points	Specifies the number of interpolation points (up to 1000) to create in the U direction; this number must be greater than the U degree number
Degree	Indicates the flexibility of the surface in the U direction, from 1 to 28; a larger number results in a more variable surface
V Direction	
Number of Points	Specifies the number of interpolation points (up to 1000) to create in the V direction; this number must be greater than the V degree number
Degree	Indicates the flexibility of the surface in the V direction, from 1 to 28; a larger number results in a more variable surface

4. Click **OK** to create the interpolated surface. Select **Show Vertices** in the Object Info palette to display the interpolation points. Use the **Reshape** tool to reshape the interpolated NURBS surface.



Interpolated NURBS Surfaces
Converting to an Interpolated Surface
Reshaping NURBS Surfaces

NURBS Curves and Surfaces

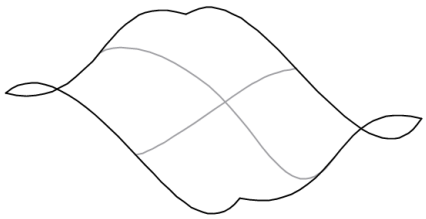
Converting to an Interpolated Surface

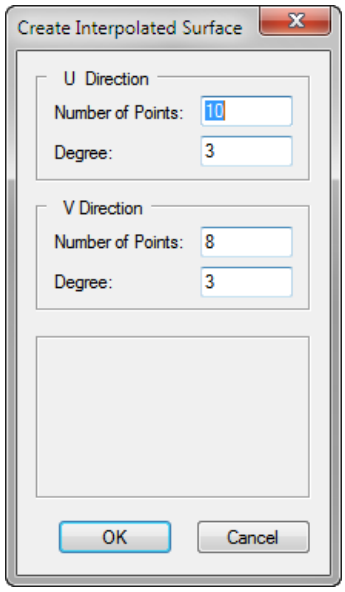
An existing untrimmed NURBS surface can be converted to an interpolated NURBS surface for easier reshaping.

To convert an untrimmed NURBS surface to an interpolated surface:

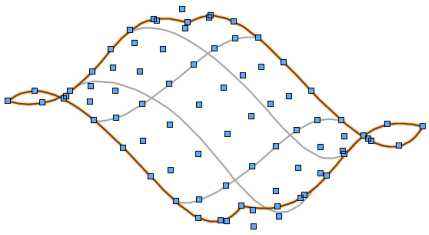
1. Select an untrimmed NURBS surface.
2. Select **Model > 3D Power Pack > Create Interpolated Surface**.

   The Create Interpolated Surface dialog box opens, with a suggested number of interpolation points and degree values for the conversion. These parameters can be changed.





3. Click **OK** to create the interpolated surface. Select **Show Vertices** in the Object Info palette to display interpolation points. Use the **Reshape** tool to reshape the interpolated NURBS surface.

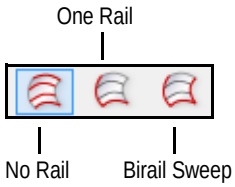


Creating an Interpolated Surface
Converting to an Interpolated Surface
Reshaping NURBS Surfaces
NURBS Curves and Surfaces

Creating a Loft Surface

The **Loft Surface** tool creates complex shapes from two or more NURBS curve cross sections with no rail, with a rail and one or more cross sections, or with two rails and one cross section. A rail is a guide NURBS curve which determines the resulting shape, in the one rail mode. In Birail Sweep mode, the rails do not need to intersect the cross sections.

Three modes are available.



Mode	Description
No Rail	Creates a loft surface using two or more NURBS curve cross sections
One Rail	Creates a loft surface using a NURBS curve rail and one or more NURBS curve cross sections
Birail Sweep	Creates a loft surface using two NURBS curve rails and one NURBS curve cross section

Similar to the **Multiple Extrude** command, the **Loft Surface** tool normally creates a 3D object (a generic solid) from a series of other objects (in this case, NURBS curves). Unlike **Multiple Extrude**, the cross sections do not need to be equally spaced, and the resulting profile can be manipulated to avoid self-intersection and to control twist.

This tool can potentially create surfaces which cannot be further manipulated in the 3D Power Pack. See "Surface Geometry Requirements" on page 327.

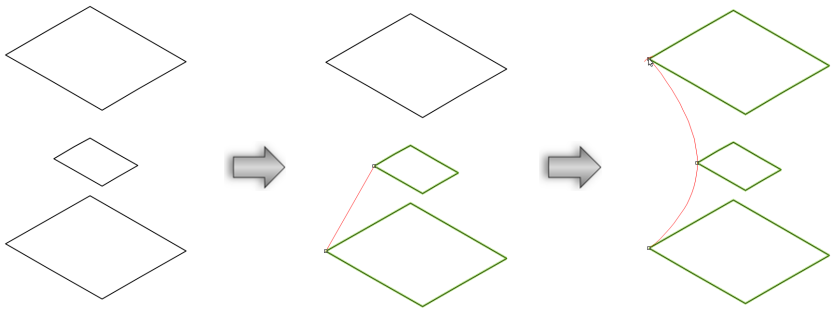
Click here for a video tip about this topic (Internet access required).

Loft Surface Using No Rails
Loft Surface Using One Rail
Loft Surface Using Two Rails
NURBS Surface Properties
Creating a NURBS Surface by Revolving a Profile Along a Rail
NURBS Curves and Surfaces

Loft Surface Using No Rails

To create a loft surface from two or more NURBS curve cross sections:

1. Click the **Loft Surface** tool from the 3D Modeling tool set, and then select **No Rail** from the Tool bar.
2. Click on each cross section. The cursor changes into an arrowhead when over a valid cross section. The point nearest to the click is selected.



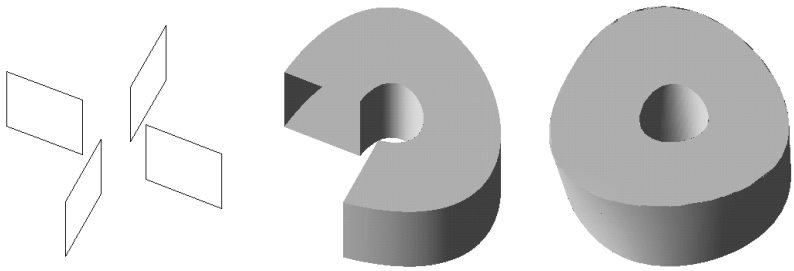
The proposed curve alignment displays in red as each section is selected.

Cross sections are created using NURBS curves which do not need to be the same shape or planar. All NURBS curves must be either closed or open objects, not a mixture of the two. 3D loci can be used in a loft, but must display at the ends of the loft.

3. Once all curves are selected, press Enter or click the check mark button on the Tool bar.

The Loft Creation dialog box opens. Specify the desired loft creation settings.

Click to show/hide the parameters.

Parameter	Description
Select Curve	Selects the previous or next cross section curve. **Reverse** changes the direction of the curve to twist or untwist the loft surface.
Reverse	Click to reverse the curve direction
Select Alignment by Point	Selects the previous or next point on the selected cross-section curve. If the cross section does not have any corners, this option is disabled.
Select Alignment by Percentage	Selects points along a geometrically continuous cross section (such as a circle) by a specified percentage, or by adjusting the slider
Ruled	Creates a linearly interpolated object 
Closed	Creates a loft surface that closes in on itself. Because the start point cannot be selected as the end point of a loft, the **Closed** option automatically completes the connection. 
Create Solid	Creates a solid loft surface with capped ends; deselect to create a group of NURBS surfaces instead of a generic solid 
Keep Curves	Retains the cross sections after the loft is created
Preview	Displays the proposed loft surfaces based on the current settings

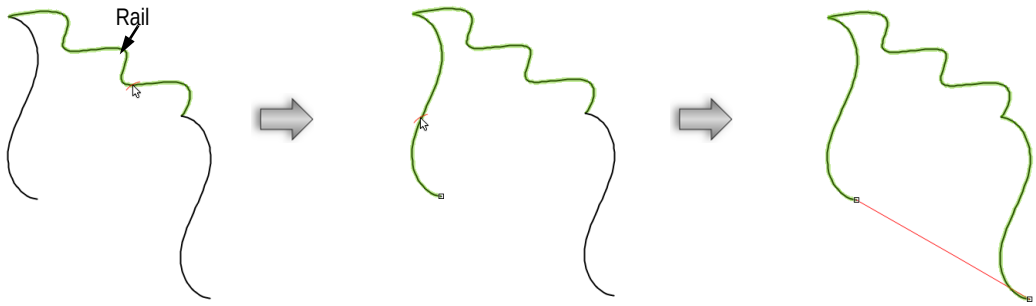
4. Click **OK** to close the dialog box and create the loft surface(s).

Creating a Loft Surface

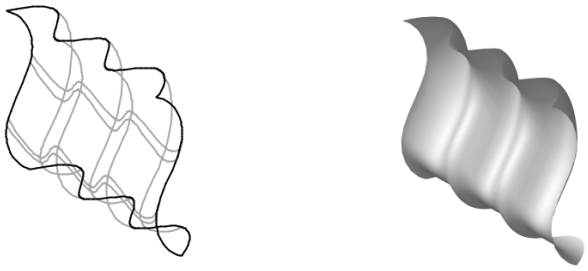
Loft Surface Using One Rail

To create a loft surface using one NURBS curve rail:

1. Click the **Loft Surface** tool from the 3D Modeling tool set, and then select **One Rail** from the Tool bar.
2. Click on the rail, and then click on each cross section.

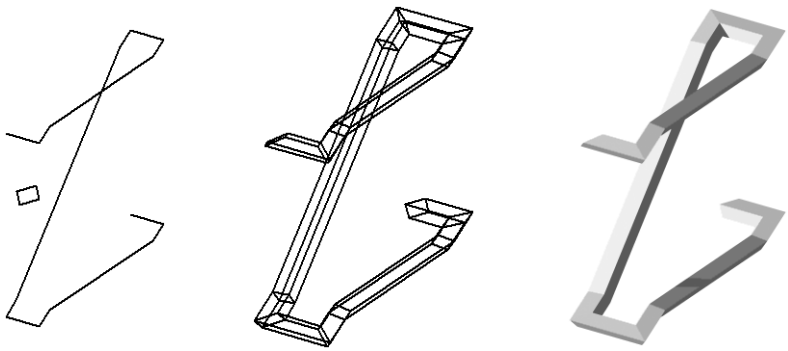


3. Once all curves are selected, press Enter or click the check mark button on the Tool bar.
   The Loft Creation dialog box opens.
4. Specify the desired loft creation settings (see “Loft Surface Using No Rails” on page 332).
5. Click **OK** to close the dialog box and create the loft surface(s).



The rail NURBS curve should intersect all cross-section curves if there is more than one curve.

When a single cross section is used, the curve is swept along the rail and the cross section does not need to intersect the rail.



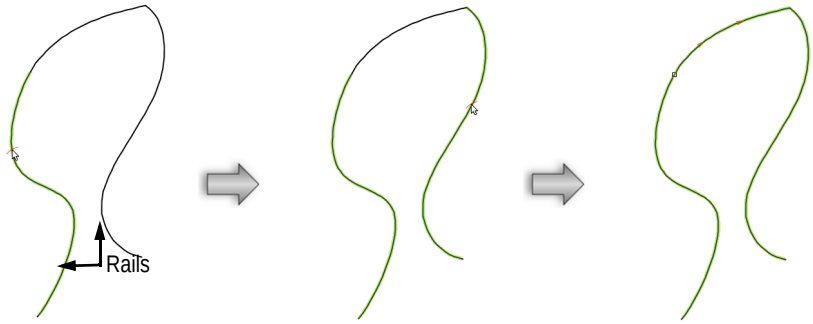
Creating a Loft Surface

Loft Surface Using Two Rails

To create a loft surface using two NURBS curve rails:

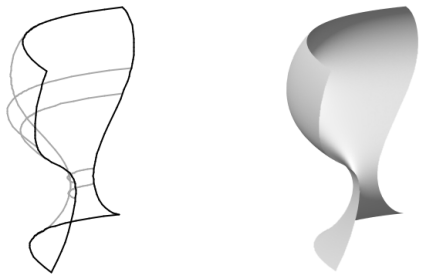
1. Click the **Loft Surface** tool from the 3D Modeling tool set, and then select **Birail Sweep** from the Tool bar.
2. Click on each rail, and then click on the cross section. The cross-section profile curve does not need to intersect the rail curves.

   In this mode, only one cross section can be selected.

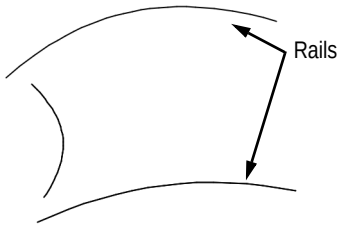
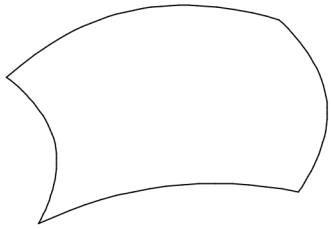
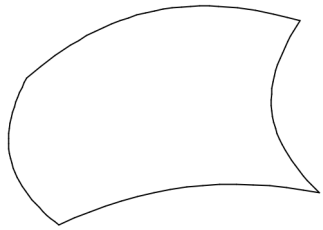
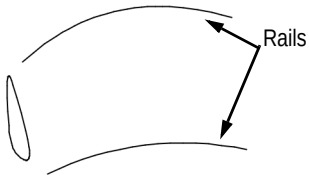
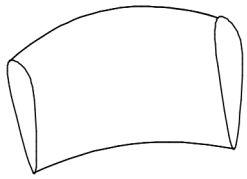
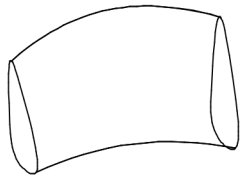


3. Once the profile curve is selected, press Enter or click the check mark button on the Tool bar.

   The Loft Creation dialog box opens.
4. Specify the desired loft creation settings (see “Loft Surface Using No Rails” on page 332).
5. Click **OK** to close the dialog box and create the loft surface(s).



The loft surface is created differently depending on whether the profile curve is open or closed, and how the start/end of the rails touch the profile curve. In certain instances, the click order (which rail is clicked first) and location (part of the profile curve that is clicked) will produce a different loft surface.

Status of Profile Curve and Rails	Effect of Rail Click Order/Profile Click Location
Open Profile Curve	
Start/end of one of the rails touches the start/end of the open profile	The rail click order and profile click location do not affect the loft surface creation
Start/end of the rails do not touch the start/end of the open profile, or do not touch the open profile at all  Rails do not touch the open profile curve	Both the rail click order and the location where the profile is clicked affect how the loft surface is created  Top rail clicked first, then bottom rail, and then the top of the profile curve OR Bottom rail clicked first, then top rail, and then the bottom of the profile curve  Top rail clicked first, then bottom rail, and then the bottom of the profile curve OR Bottom rail clicked first, then top rail, and then the top of the profile curve
Closed Profile Curve	
Start/end of one of the rails touches the closed profile curve	The click order and location do not affect the loft surface creation
Start/end of the rails do not touch the closed profile at all  Rails do not touch the closed profile curve	The click order affects how the loft surface is created  Top rail clicked first, then bottom rail, and then the profile curve  Bottom rail clicked first, then top rail, and then the profile curve

Creating a Loft Surface

Creating Surfaces from Curves

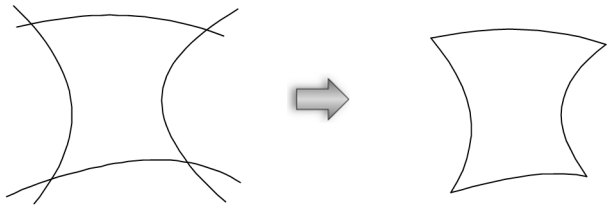
The **Create Surface from Curves** command creates a NURBS surface from a network of single closed curves (including closed 2D objects) or two or more open NURBS curves. These NURBS curves can be planar or non-planar, but NURBS in the U direction must intersect NURBS in the V direction at only one point.

> This command can potentially create surfaces which cannot be further manipulated in the 3D Power Pack. See "Surface Geometry Requirements" on page 327.

To create a NURBS surface from curves:

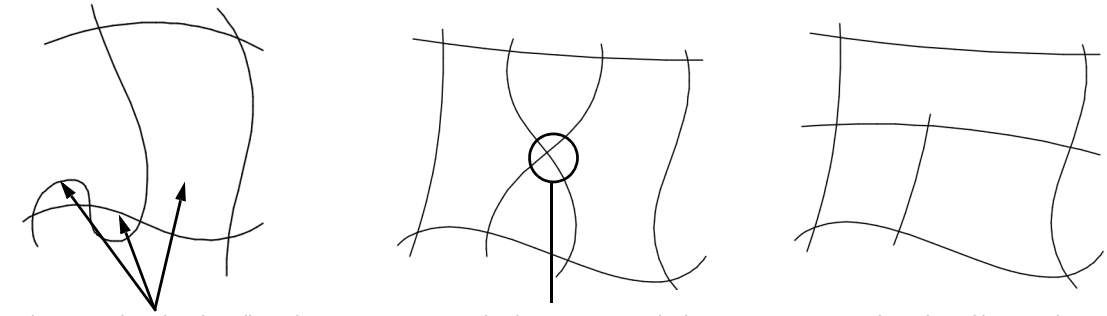
1. Draw two or more open NURBS curves to create an enclosed region.
2. Select the NURBS objects being used to create the NURBS surface.
3. Select **Model > 3D Power Pack > Create Surface from Curves**.

   A NURBS surface is created.



Multiple open NURBS that form both boundary and interior curves can be used to create a NURBS surface, but must have an equal number of intersections.

A NURBS surface is not created if there is more than one intersection created by the selected curves, creating multiple enclosed regions. It is also not created if there is an intersection between selected curves in the same parametric direction or if there are an unequal number of intersections.



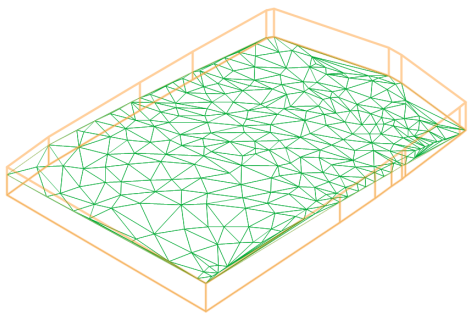
Creating a Loft Surface
Creating a Drape Surface
Creating a Fillet Surface
Creating Planar Caps
Creating a NURBS Surface by Revolving a Profile Along a Rail
NURBS Surface Properties
NURBS Curves and Surfaces

Creating a Drape Surface

You can automatically create a rectangular, draped NURBS surface over an object, up to a specific Z plane value. This is useful for representing surfaces including curtains, tablecloths, product packaging, and (with Vectorworks Architect or Landmark) site models.

To create a drape surface:

1. Select one or more objects to be draped.



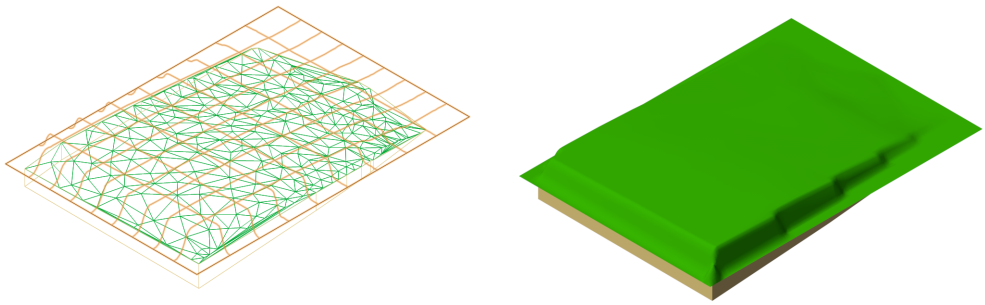
2. Select **Model > 3D Power Pack > Create Drape Surface**.

   The Create Drape Surface dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Number of Points	Specifies the number of control points on the drape surface; the greater the number of points, the closer the drape surface is to the object
U Direction	Enter the number of points to create in the U direction; this number must be greater than 3
V Direction	Enter the number of points to create in the V direction; this number must be greater than 3
Plane Z Value	Specify the base level of the drape surface (how far down the bottom of the drape goes), which must be less than the highest Z coordinate of the object

3. Specify the desired parameters, and click **OK** to create the drape surface.



Click here for a video tip about this topic (Internet access required).

Creating a Loft Surface
Creating Surfaces from Curves
Creating a Fillet Surface
Creating Planar Caps
Creating a NURBS Surface by Revolving a Profile Along a Rail
NURBS Surface Properties
NURBS Curves and Surfaces

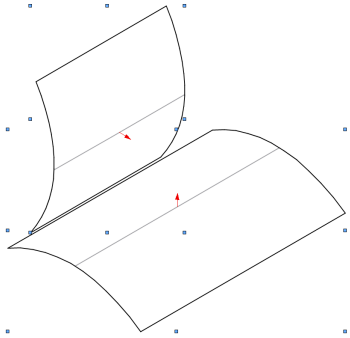
Creating a Fillet Surface

The **Create Fillet Surface** command creates a surface between two selected surfaces, trimming the original surfaces by default.

This command is useful for sheet metal design (for example, for car bodies) to provide a smooth transition between two adjacent surfaces—perhaps for aesthetic quality or because of a manufacturing requirement.

To create a fillet surface at the intersection of two selected NURBS surfaces:

1. Select two NURBS surfaces. The position of the fillet surface depends on the normal of the NURBS surfaces; select **Show Normal** in the Object Info palette to display the surface normals (see “Displaying Surface Normals” on page 326).

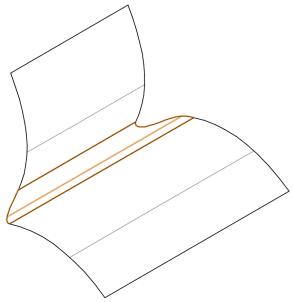
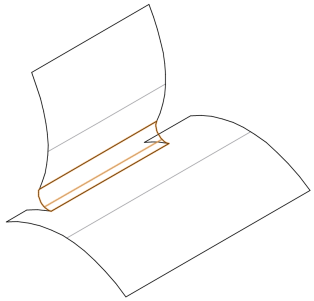


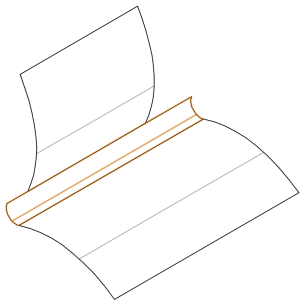
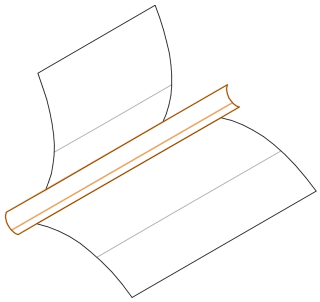
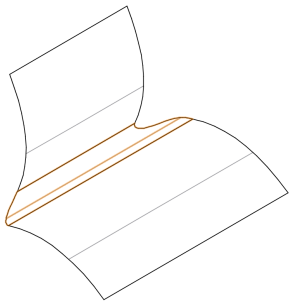
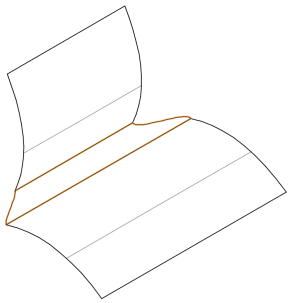
2. Select **Model > 3D Power Pack > Create Fillet Surface**.

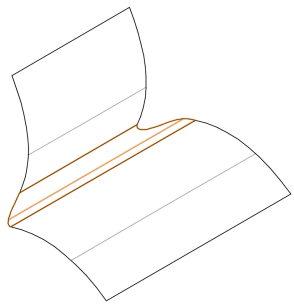
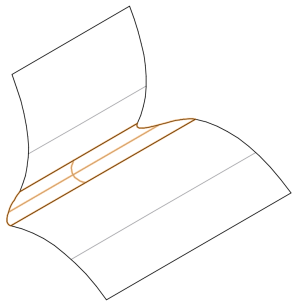
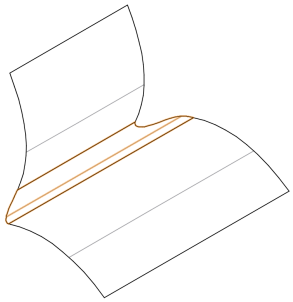
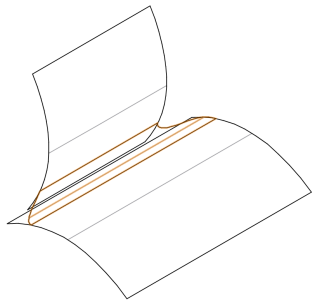
   The Fillet Surface Preferences dialog box opens. Enter the desired **Fillet Radius**, and then select the trim and cross section type for the fillet.

   The following trim type examples use a circular cross section type, and the cross section examples use an edge tangency trim type. The original surfaces are trimmed.

Click to show/hide the parameters.

Parameter	Description
Fillet Radius	Specifies the radius of the fillet surface
Trim Type	
Edge Tangency	Creates a blend between the minimal and maximal intersections on each end of the fillet 
Minimal	Trims the fillet surface to the minimal intersecting areas on the face boundaries 

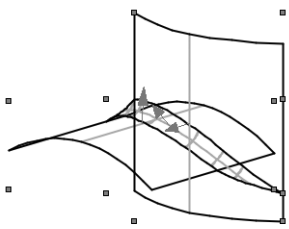
Parameter	Description
Maximal	Trims the fillet surface to the maximal intersecting areas on the face boundaries 
None	Generates the entire fillet without trimming the fillet surface 
Cross Section Type	
Circular	Creates a fillet surface with a circular cross section 
Linear	Creates a fillet surface with a linear cross section 

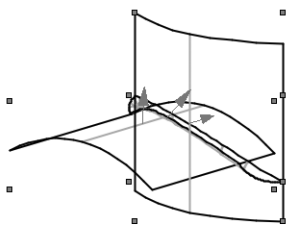
Parameter	Description
Tangent Continuous Blend	Creates a fillet surface with a tangent continuous G1 cross section 
Curvature Continuous Blend	Creates a fillet surface with a curvature continuous G2 cross section 
Trim Original Surfaces	Creates a fillet surface and trims the original surfaces where they connect to the fillet surface  Trim Original Surfaces selected  Trim Original Surfaces deselected

3. Click **OK**.

   A NURBS fillet surface is created between the two objects.

   The position of the fillet surface depends on the surface normals. The fillet is created differently if the **Reverse Normal** button is clicked for one or both of the surfaces in the Object Info palette.





Fillet surface after reversing the normal of the vertical surface

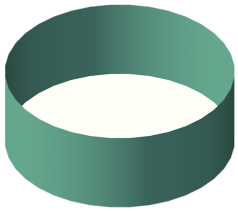
Creating a Loft Surface
Creating Surfaces from Curves
Creating a Drape Surface
Creating Planar Caps
Creating a NURBS Surface by Revolving a Profile Along a Rail
NURBS Surface Properties
NURBS Curves and Surfaces

Creating Planar Caps

The **Create Planar Caps** command closes off the ends of open-ended solids by creating planar NURBS surfaces. These surfaces can then be added to the rest of the surfaces with the **Add Solids** command.

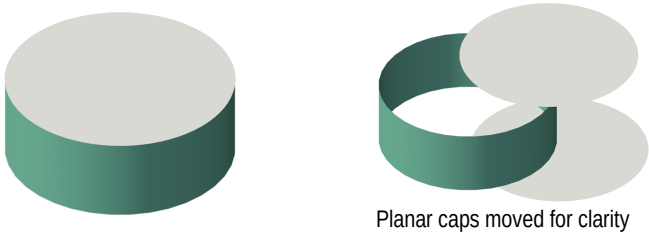
To create planar caps:

1. Select the open-ended solid.



2. Select **Model > 3D Power Pack > Create Planar Caps**.

   Planar NURBS surfaces are created to close off the ends of the solid.



Planar caps moved for clarity

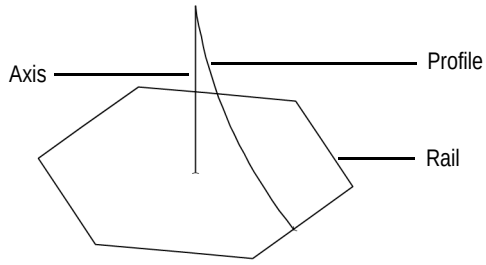
3. If desired, add the solid and the planar cap surfaces by selecting them all and choosing **Model > Add Solids**.

Creating a Loft Surface
Creating Surfaces from Curves
Creating a Drape Surface
Creating a Fillet Surface
Creating a NURBS Surface by Revolving a Profile Along a Rail
NURBS Surface Properties
NURBS Curves and Surfaces

Creating a NURBS Surface by Revolving a Profile Along a Rail

Complex NURBS surfaces can be created by revolving a profile along a guide curve (rail). This is an alternate, and sometimes easier, method of creating NURBS surfaces than using the **Loft Surface** tool, especially when creating tent-like structures.

The **Revolve with Rail** command creates the NURBS surface by revolving a planar NURBS curve about an axis. The revolution is guided by a rail curve on a plane perpendicular to the plane containing the profile curve and axis.



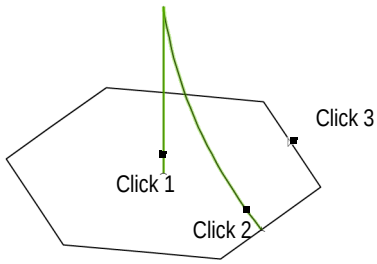
To create a NURBS surface with profile and rail:

1. Create the axis, rail and profile out of NURBS curves.

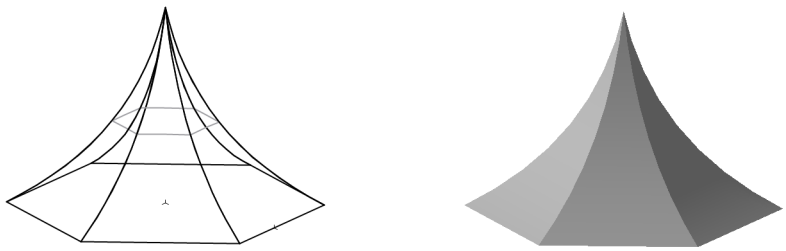
   Conditions must meet the following requirements:

   - The axis must be a linear NURBS curve
   - The profile must be a planar NURBS curve
   - The profile cannot intersect the axis, though it can touch
   - The axis must lie on the same plane as the profile
   - The rail must be a planar NURBS curve that lies on a plane perpendicular to the plane containing the axis and profile

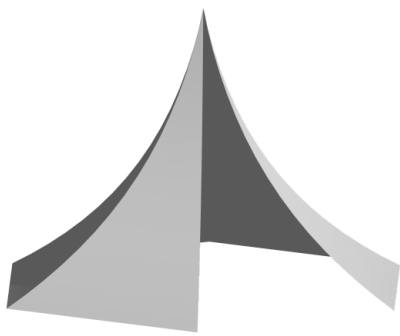
2. Select **Model > 3D Power Pack > Revolve with Rail**. Select, in order, the axis, profile, and rail.



3. The NURBS surfaces are automatically created.



   The rail defines the extent of the revolution. An open rail curve generates surfaces until the rail ends.



Creating a Loft Surface
Creating Surfaces from Curves
Creating a Drape Surface
Creating a Fillet Surface
Creating Planar Caps
NURBS Surface Properties
NURBS Curves and Surfaces

NURBS Surface Properties

The properties of a NURBS surface are displayed in the Object Info palette, where they can also be edited. NURBS surfaces can also be reshaped with the **Reshape** tool; see “Reshaping Objects” on page 1043 for more information.

Click to show/hide the parameters.

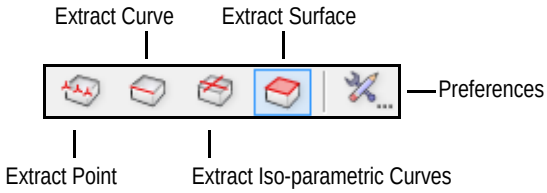
Parameter	Description
Move	Select **Entire Object** to edit all the vertices of the surface, **Vertex Only** to edit only the selected vertex, **U Vertices** to edit all of the vertices in a row in the U direction, and **V Vertices** to edit all of the vertices in a row in the V direction
Edit U/V	Scrolls through the vertices of the selected NURBS surface in either the U parametric direction or V parametric direction
X, Y, Z / X’,Y’,Z’	Depending on the selection in the **Move** list, displays the position of the current surface or vertex; edit the values to change the surface/vertex position
U Degree/ V Degree	A NURBS surface is defined mathematically by two parameters, U and V, which increase or decrease along certain directions called the U and V parametric directions. The two directions are perpendicular to each other. Increasing the U or V Degree value adds vertices in that parametric direction; the vertices can then be manipulated.
Weight	NURBS curves and surfaces are represented mathematically by weighted control points. The weight value can be anywhere between .01 and 100. A weight above 1 pulls the curve or surface toward the control point; a weight below 1 has the reverse effect.
Reverse Normal	Flips the surface normal direction
Untrim	Creates a NURBS surface, if the surface has been trimmed
NURBS Surface parameters (display only)	Displays the NURBS surface parameters
Show Vertices	Select to display the surface’s vertices
Show Normal	Displays the surface normal as a red arrow for the selected NURBS surface

NURBS Curves and Surfaces
Creating a Loft Surface
Creating Surfaces from Curves
Creating a Drape Surface
Creating a Fillet Surface
Creating Planar Caps
Creating a NURBS Surface by Revolving a Profile Along a Rail

3D Power Pack Cursors

Extracting Geometry

The **Extract** tool can be used to extract geometry from the edge or surface of a NURBS surface or solid object, leaving the original unmodified. The extracted geometry can then be used for snapping or for other surface operations.



Mode	Description
Extract Point	Extracts 3D loci from the edges of a NURBS surface or solid object. 3D loci are placed at the start, end, and mid-points of the selected edges. For circular edges, a 3D locus is created at the circle center as well.
Extract Curve	Extracts a NURBS curve from the edge of a solid object
Extract Iso-parametric Curves	Extracts an iso-parametric curve from the surface of a solid object
Extract Surface	Extracts a NURBS surface from the face of a solid
Preferences	Provides options for extracting tangent entities, faces, or all entities; the specific items extracted depends on which mode is selected. These options have no effect in Extract Iso-parametric Curves mode.

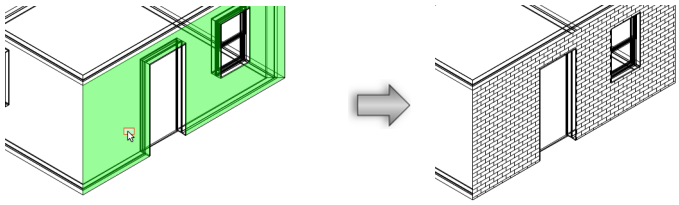
To extract geometry from the edge or surface of a NURBS surface or solid object:

1. In the desired view, click the **Extract** tool from the 3D Modeling tool set, and then select the desired mode from the Tool bar.
2. If **Extract Iso-parametric Curves** mode was selected, skip to step 4. For all other modes, click **Preferences** from the Tool bar.

   The Extract Preferences dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Select Tangent Entities	In **Extract Point** or **Extract Curve** mode, extracts loci or curves from tangentially connected edges of the selected object; in **Extract Surface** mode, extracts surfaces from tangentially connected faces of the selected object
Select Faces	In **Extract Point** or **Extract Curve** mode, extracts loci or curves from all edges of the selected face(s)
Select All Entities	In **Extract Point** or **Extract Curve** mode, extracts loci or curves from all edges of the selected object; in **Extract Surface** mode, extracts surfaces from all faces of the selected object

Parameter	Description
Create Planar Objects	In **Extract Surface** mode, creates planar objects from faces by extracting the geometry of a planar face. For example, create a polyline extracted from the face of a wall. The polyline could then have a hatch applied to simulate the appearance of brick in 3D views.  Another way to create planar objects from faces is with the Polygon from Inner Boundary mode of the **2D Polygon** tool.

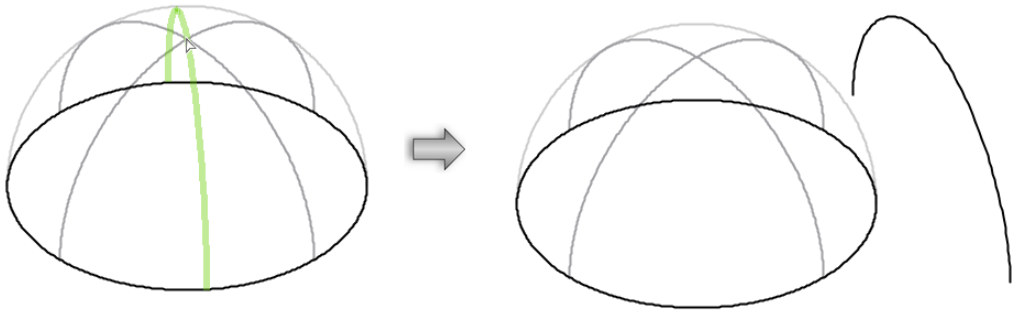
3. Specify the desired preferences and click **OK**.

   Select the edge(s) or surface(s) from which geometry will be extracted. To select multiple edges or surfaces, hold the Shift key while selecting. To select the back faces of solids, press the Alt key (Windows) or Option key (Mac).

   See “Selecting the Edges and Faces of a Solid” on page 324 for information on selecting surfaces.

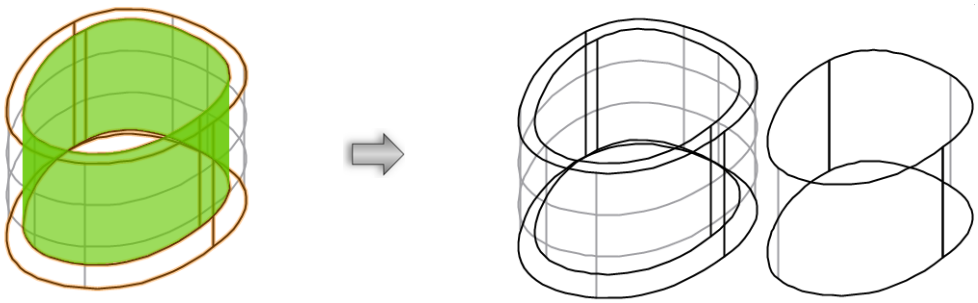
4. Press Enter or click the check mark button on the Tool bar to extract the 3D loci, curves, or surfaces.

   To edit extracted groups, select **Modify > Ungroup**.



In Extract Iso-parametric Curve mode, click the hemisphere surface to select curve(s)

Extracted curve moved for clarity



In Extract Surface mode, with the Select Tangent Entities preference selected, click an interior surface to select all tangent interior surfaces

Extracted surface moved for clarity

Creating a Polygon from an Inner Boundary
NURBS Curves and Surfaces

Analyzing NURBS Curves and Surfaces

The **Analysis** tool provides proximity, intersection, and curvature information for NURBS curves and surfaces.

Two modes are available.

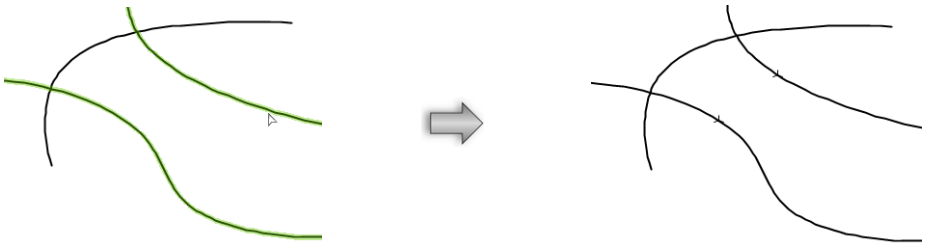
Mode	Description
Proximity	Determines the minimum distance between NURBS curves/surfaces and 3D loci, or the intersection between NURBS curves/surfaces
Interrogation	Interactively determines the curvature of NURBS curves and surfaces and displays curvature parameters

Determining NURBS Curve and Surface Proximity

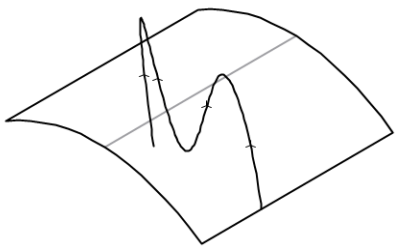
In **Proximity** mode, the **Analysis** tool places a 3D locus at the minimum distance or intersection between two NURBS curves, a NURBS curve and NURBS surface, and a 3D locus and NURBS curve or surface. The tool places a NURBS curve at the intersection of two NURBS surfaces.

To analyze NURBS proximity or intersection:

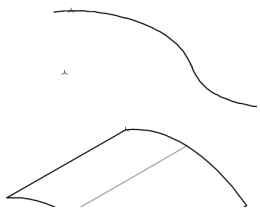
1. Click the **Analysis** tool from the 3D Modeling tool set, and then select **Proximity** from the Tool bar.
2. Click on a NURBS curve, NURBS surface, or 3D locus, and then click on another NURBS curve or surface.



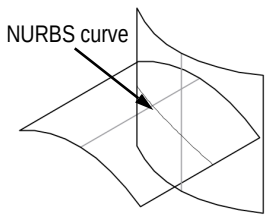
A 3D locus is placed on each NURBS curve at the closest minimum distance or intersecting points between the two items, or a NURBS curve is placed at the intersection of two NURBS surfaces.



Locus points placed at each intersection between a NURBS curve and NURBS surface



Locus points placed at the minimum distance from a 3D locus point to both a NURBS curve and a NURBS surface



NURBS curve placed at the intersection of two NURBS surfaces

Determining NURBS Curvature, Tangents, and Normals

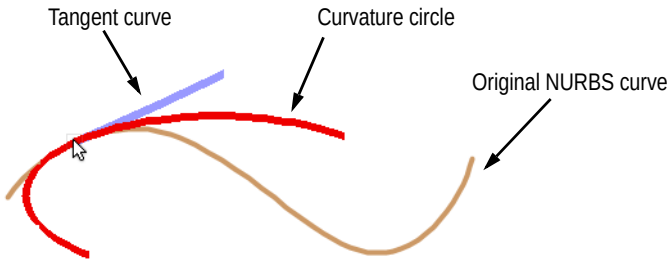
In **Interrogation** mode, the **Analysis** tool provides curvature parameters for NURBS curves and NURBS surfaces. It also shows curvature circles and tangent and normal curves interactively, for any point on the existing curve or surface.

Optionally, you can add these curves to the drawing; the tool can also place two 3D loci for each curvature circle: one locus where the circle meets the NURBS curve or surface, and one at the center of the circle.

Analyzing NURBS Curves

To determine the curvature of a NURBS curve, or create a tangent curve:

1. Click the **Analysis** tool from the 3D Modeling tool set, and then select **Interrogation** from the Tool bar.
2. Click on the NURBS curve of interest, and then move the cursor along the curve to display the curvature circle (in red) and tangent curve (in blue) at the cursor position.

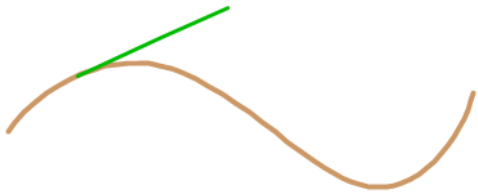


3. Click the curve to obtain curve properties at a specific location along the curve. The Curve Properties dialog box opens, displaying curvature parameters for that location.

Click to show/hide the parameters.

Option	Description
Create Curvature Circle	Adds the NURBS curve displaying in red to the drawing
Locus at center of curvature	Adds a 3D locus at the point where the curvature circle meets the NURBS curve, and another locus at the center of the circle
Create Tangent Curve	Add the NURBS curve displaying in blue to the drawing
Length	Specifies the length of the tangent curve

4. To add a curvature circle or tangent curve to the drawing, select the appropriate options and click **OK**. If you select multiple items, they are created as a group.

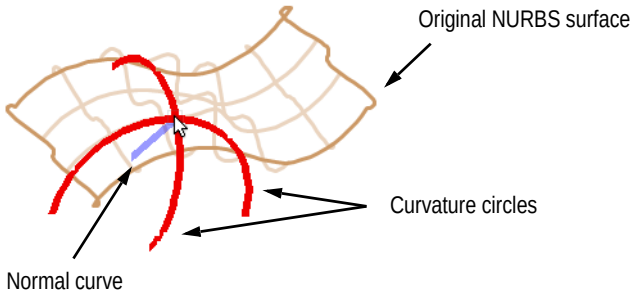


Click a point on the existing NURBS curve and select **Create Tangent Curve** to place a curve at that point

Analyzing NURBS Surfaces

To determine the curvature of a NURBS surface, or create a normal curve:

1. Click the **Analysis** tool from the 3D Modeling tool set, and then select **Interrogation** from the Tool bar.
2. Click on the NURBS surface of interest and move the cursor along the surface to display the curvature circles (in red) and normal curve (in blue) at the cursor position.

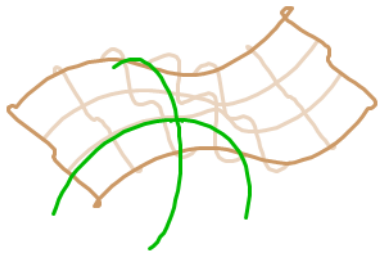


3. Click again to obtain surface properties at a specific location along the surface. The Surface Properties dialog box opens, displaying curvature parameters for that location.

Click to show/hide the parameters.

Option	Description
Create Curvature Circles	Adds the NURBS curve(s) displaying in red to the drawing
Loci at centers of curvature	Adds a 3D locus at the point where the curvature circle(s) meet the NURBS surface, and another locus at the center of each circle
Create Normal Curve	Add the NURBS curve displaying in blue to the drawing
Length	Specifies the length of the normal curve

4. To add curvature circle(s) or a normal curve to the drawing, select the appropriate options and click **OK**. If you select multiple items, they are created as a group.



Click a point on the existing NURBS surface and select Create Curvature Circles to place curves at that point

NURBS Curves and Surfaces

Rebuilding NURBS Curves and Surfaces

The number of vertices in one or more selected NURBS curves or untrimmed surfaces can be changed with the **Rebuild NURBS** command. Reducing vertices simplifies the geometry, making it easier to manipulate, and increases the speed and ease-of-use for other objects that are based upon it.

To rebuild a NURBS curve or surface:

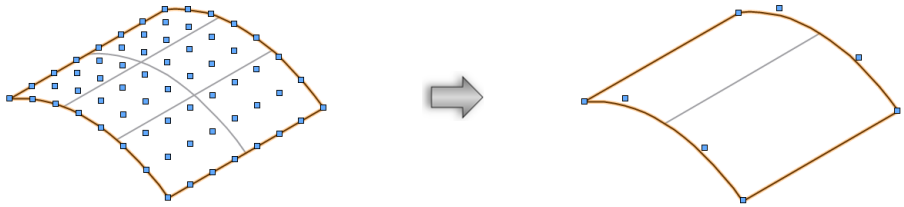
1. Select the NURBS curve(s) or surface(s).
2. Select **Model > 3D Power Pack > Rebuild NURBS**.

   The Rebuild NURBS dialog box opens. Enter the number of points to use; specifying fewer points simplifies the NURBS curve or surface, but increases the difference between the original geometry and the rebuilt geometry.

Click to show/hide the parameters.

Parameter	Description
Number of Points	Specifies the number of points to use; for a NURBS curve, enter the number of points, which must be at least 3
U Direction	For a NURBS surface, enter the number of points to use along the U direction; this number must be at least 3
V Direction	For a NURBS surface, enter the number of points to use along the V direction; this number must be at least 3
Keep Original	Retains the original curve or surface, and adds a new, rebuilt curve or surface
Maximum Deviation	After clicking **Preview**, displays the maximum deviation between the original and rebuilt surface or curve
Preview	Displays a preview of the new, rebuilt curve or surface

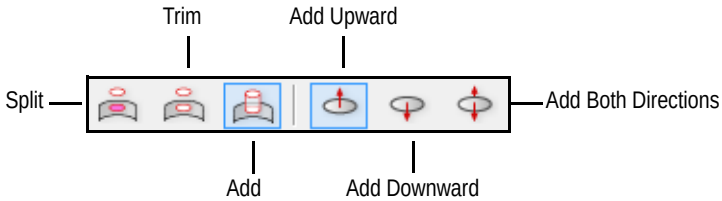
3. Click **OK** to rebuild the NURBS curve(s) or surface(s).



NURBS Curves and Surfaces

Splitting or Trimming NURBS Surfaces

The **Project** tool projects a 2D object or NURBS curve onto a NURBS surface, and then splits the NURBS surface or trims the NURBS surface according to the region selected. This allows nonlinear splitting or trimming of surfaces to create different shapes or cut a hole in an object.



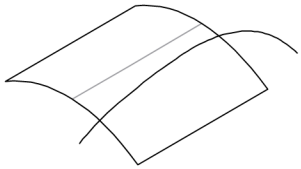
Mode	Description
Split	Splits a NURBS surface with a projection
Trim	Trims a NURBS surface with a projection
Add	Adds a projection to a NURBS surface
Add Upward	In **Add** mode, adds the projection in the profile plane normal direction
Add Downward	In **Add** mode, adds the projection in the opposite direction from the profile plane normal direction
Add Both Directions	In **Add** mode, adds the projection in both the profile plane normal direction and the opposite direction

Project and Split
Project and Trim
Project and Add
Creating a Rib
Splitting Objects and NURBS Surfaces with the Split Tool
NURBS Curves and Surfaces

Project and Split

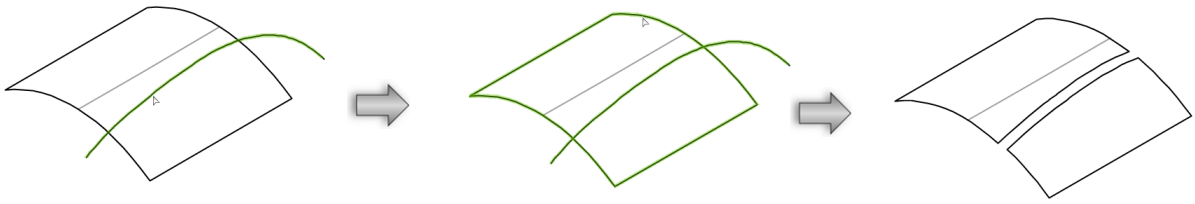
To split a NURBS surface with a projection:

1. In the desired view, draw a 2D object or NURBS curve on top of a NURBS surface.



If the object to be projected is an open NURBS curve or open 2D object, both end points must be outside the NURBS surface.

2. Click the **Project** tool from the 3D Modeling tool set, and then select **Split** from the Tool bar.
3. Click on the splitting object, and then click on the NURBS surface.



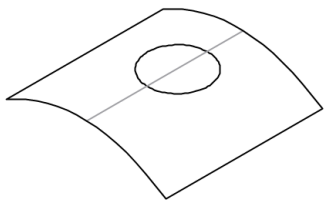
The splitting object is projected onto the NURBS surface, resulting in a group of NURBS surfaces.

Selecting the Edges and Faces of a Solid
Splitting or Trimming NURBS Surfaces
Splitting Objects and NURBS Surfaces with the Split Tool

Project and Trim

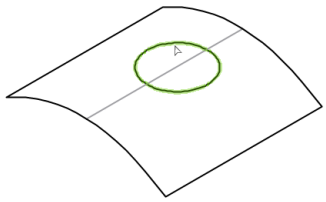
To trim a NURBS surface with a projection:

1. In the desired view, draw a 2D object or NURBS curve on top of a NURBS surface.

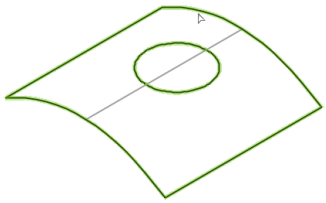


If the object to be projected is an open NURBS curve or an open 2D object, both end points must be outside the NURBS surface.

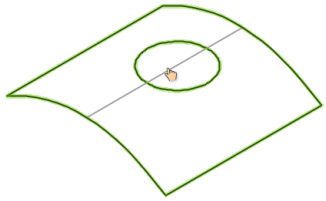
2. Click the **Project** tool from the 3D Modeling tool set, and then select **Trim** from the Tool bar.
3. Click on the trimming object, and then click on the NURBS surface. The cursor changes into the pointing hand cursor. Click on the side of the intersection to be trimmed away.



Select the project and trimming object



Select the NURBS surface



Select the region to be trimmed

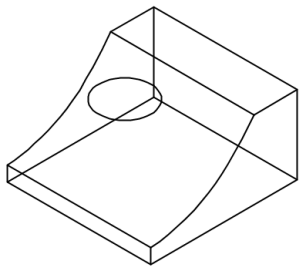
The trimming object is projected onto the NURBS surface, and the region selected to be trimmed away is removed from the NURBS surface at the point of intersection, resulting in a group of NURBS surfaces.

Selecting the Edges and Faces of a Solid
Splitting or Trimming NURBS Surfaces

Project and Add

To add a projection to a NURBS surface:

1. In the desired view, draw a 2D object or NURBS curve on top of a NURBS surface.

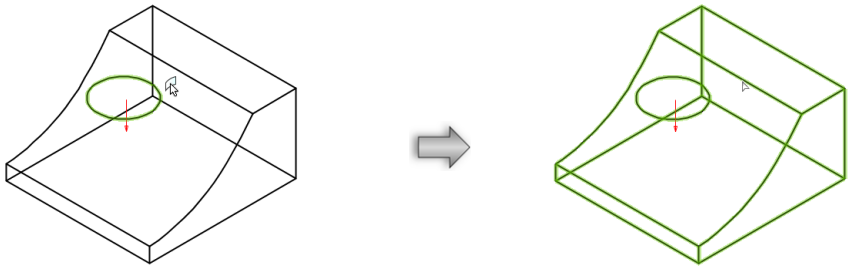


The 2D object or NURBS curve must be closed and planar.

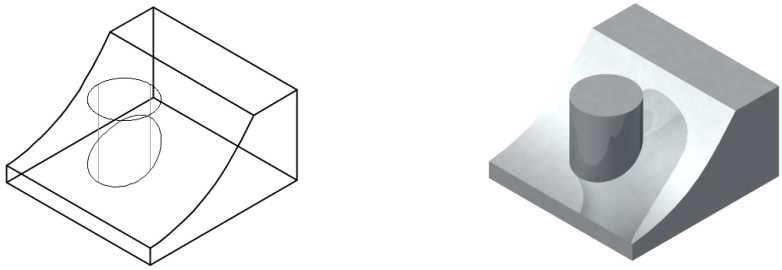
2. Click the **Project** tool from the 3D Modeling tool set.
3. Select **Add** from the Tool bar. Select the appropriate direction mode so that the projection direction intersects the NURBS surface (**Add Upward**, **Add Downward**, or **Add Both Directions**).

The projection direction must intersect with the NURBS surface, or no projection is created.

4. Click on the object to add; the projection direction is indicated by a red arrow. Click on the NURBS surface.



The object's projection is added to the NURBS surface at the point of intersection, creating a generic solid.



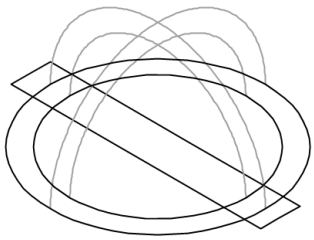
Selecting the Edges and Faces of a Solid
Splitting or Trimming NURBS Surfaces

Creating a Rib

The **Add** mode of the **Project** tool can be used to create ribs.

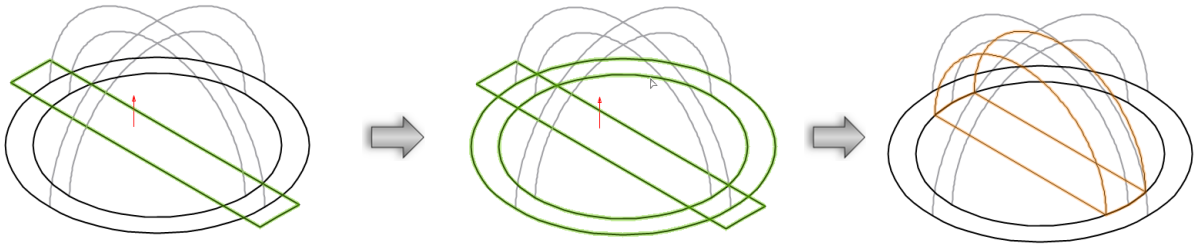
To create a rib:

1. In the desired view, draw a 2D object or NURBS curve on top of a solid.



The 2D object or NURBS curve must be closed and planar.

2. Click the **Project** tool from the 3D Modeling tool set, and then select **Add** from the Tool bar.
3. Select the appropriate projection direction mode.
4. Click on the object to add, and then click on the solid.



The rib profile is projected until it meets the next surface it encounters. The profile is trimmed at the extremities automatically.

Selecting the Edges and Faces of a Solid
Splitting or Trimming NURBS Surfaces

Extending NURBS Curves and Surfaces

NURBS curves and surfaces can be extended by a specified distance with the **Extend NURBS** command.

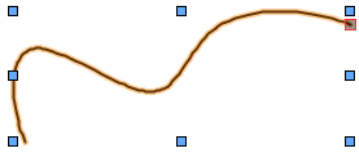
Closed NURBS curves and trimmed NURBS surfaces cannot be extended. In addition, a NURBS surface cannot be extended in the direction (U or V) that it is closed.

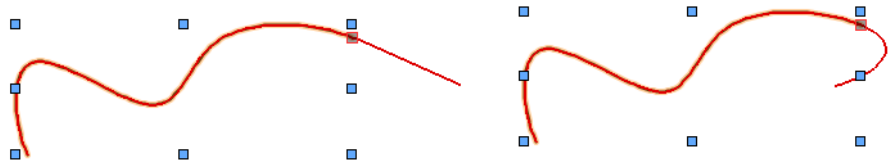
To extend a NURBS curve or surface:

1. Select the NURBS curve or surface to extend.
2. Select **Model > 3D Power Pack > Extend NURBS**.

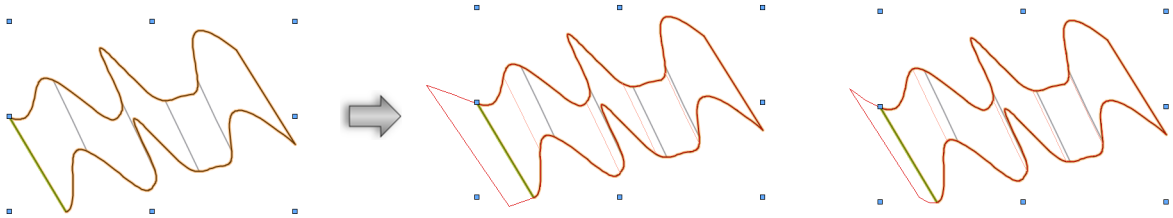
   The Extend NURBS dialog box opens. Select the end point (for NURBS curves) or edge (for NURBS surfaces) to extend. On the drawing, the selected edge or end point is displayed in red. Select the type of extension to make and specify the extension distance.

Click to show/hide the parameters.

Parameter	Description
Select End Point/Edge	Click the arrows to select the end point or edge to extend. The selected end point or edge is displayed in red on the drawing. 
Linear Extension	Extends the edge or end point tangentially
Smooth Extension	Extends the edge or end point by continuing the current curvature
Distance	Specifies the extension distance

Parameter	Description
Preview	Click to see a preview of the extension in red  Linear Extension Smooth Extension

3. Click **Preview** to check the extension, and then click **OK** to extend the NURBS curve or surface.



Selecting the Edges and Faces of a Solid
NURBS Curves and Surfaces

Creating Helix-Spirals

The **Create Helix-Spiral** command creates a helix-shaped or spiral-shaped 3D object from one or more path objects. The paths can be 2D objects or NURBS curves.

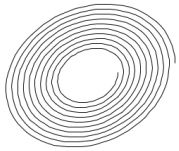
To create a helix or spiral:

1. Select the object(s) to use as the path.
2. Select **Model > 3D Power Pack > Create Helix-Spiral**.

   The Helix-Spiral Creation dialog box opens.

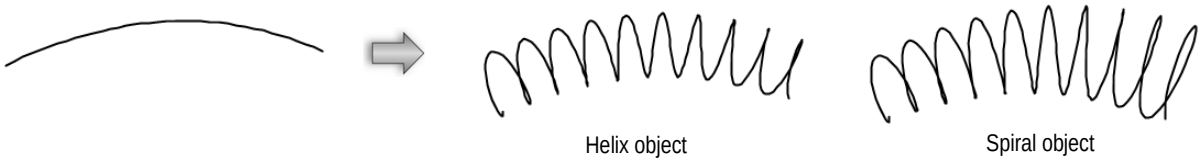
Click to show/hide the parameters.

Parameter	Description
Use Turns	Select to create a helix/spiral by a specified number of turns along the path
Turns	Enter the total number of turns to be created along the path
Use Pitch	Select to create a helix/spiral by pitch
Pitch	Enter the pitch value (the distance between successive turns)
Start Radius	Enter the radius of the perpendicular starting circle
End Radius	Enter the radius of the perpendicular ending circle. To create a helix, enter a value matching the start radius. To create a spiral, enter a different value.
Start Angle	Enter the starting point of the twist on the starting circle
Reverse Direction	Select to change the rotation direction of the helix/spiral from clockwise to counter-clockwise or vice-versa

Parameter	Description
Flatten	Select to flatten the helix/spiral into a 2D spiral. **Use Pitch** is not available with this option. 

3. Specify the object parameters, and click **OK**.

   A helix or spiral is created.



A helix or spiral object can be used, without further conversion, as a path object for the **Extrude Along Path** command.

Ungrouping a helix-spiral object results in a NURBS curve.

Creating Contours

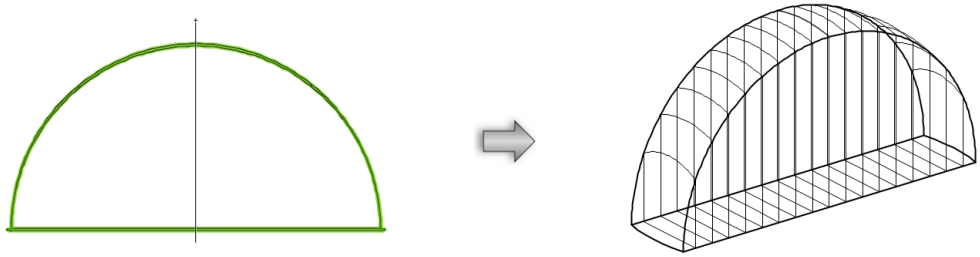
Contours are intersections of a solid or surface with a plane passing through the line specified with the **Create Contours** tool. This tool creates contours at specified intervals, which can then be used to create a loft surface in order to re-create a solid shape.

 To create contours:

1. Click the **Create Contours** tool from the 3D Modeling tool set, and then select **Preferences** from the Tool bar to specify the contour interval.

   To create a single contour, specify a **Contour Increment** of zero.
2. Click **OK**.
3. Click and drag to indicate the position of the intersecting plane.

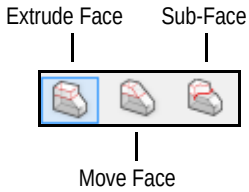
   The contours, a group of NURBS curves, are created.



Direct Modeling with the Push/Pull Tool

The **Push/Pull** tool interactively reshapes planar geometry, creating bosses (protrusions) or cutouts on solids by adding or subtracting volume from a solid. The volume is created or subtracted by extruding a face in **Extrude Face** mode, including all the geometry present on a face in **Move Face** mode, or extruding a planar NURBS curve (or group of closed, non-intersecting NURBS curves) in **Sub-Face** mode.

Three modes are available.



Mode	Description
Extrude Face	Selects the planar face of a solid, a 2D planar object, or NURBS curve, to add to or subtract volume, creating perpendicular faces
Move Face	Extends the planar face of a solid, adding or subtracting the face along with any geometry present on the face
Sub-Face	Selects a curve or group of curves to add to or subtract volume from a solid

To preserve the primitive profile geometry of an extrude, right-click (Windows) or Ctrl-click (Mac) on the extrude and select **Edit** from the context menu. See “Object Editing Mode” on page 1004.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Face-based Addition or Subtraction
Moving the Face of a Solid
Curve-based Addition or Subtraction
NURBS Curves and Surfaces

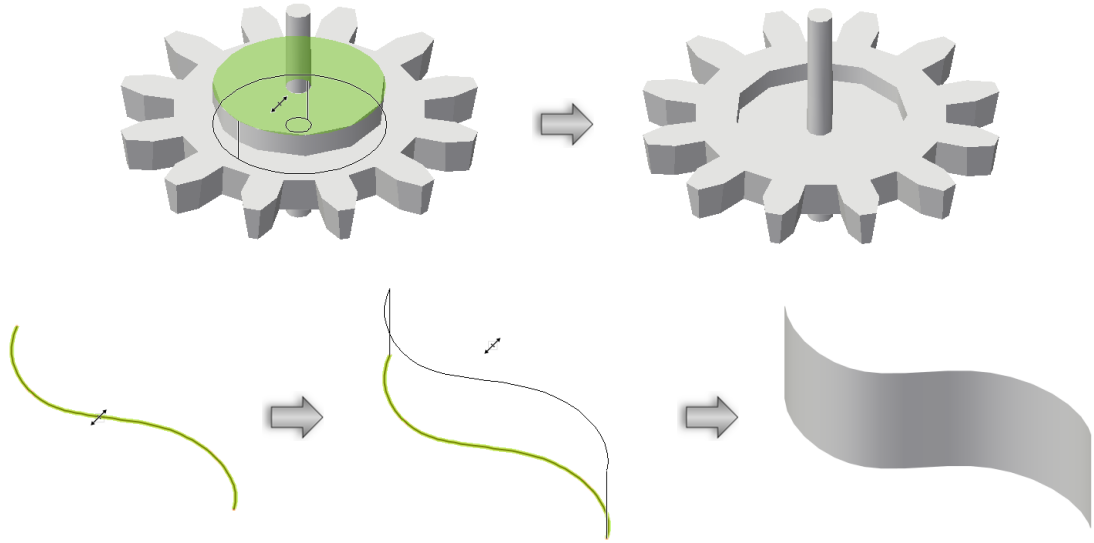
Face-based Addition or Subtraction

The **Extrude Face** mode creates perpendicular faces as volume is added or subtracted. Objects that can be modified include 2D planar objects, NURBS curves, and the planar face of any solid.

To add to or subtract from a solid in **Extrude Face** mode:

1. Click the **Push/Pull** tool from the 3D Modeling tool set, and then select **Extrude Face** from the Tool bar.
2. Click on the desired planar face, planar object, or NURBS curve.

   If needed, the Select Face dialog box opens to help select the correct face, as described in “Selecting the Edges and Faces of a Solid” on page 324. To select the back faces of solids, press the Alt key (Windows) or Option key (Mac) while selecting. When the correct face is selected, click **OK**.
3. The cursor changes to a double-headed arrow. Move the cursor to expand (**Add** mode) or decrease (**Subtract** mode) the solid volume, or create an extrusion from the 2D planar object or NURBS curve. The cursor direction determines whether volume is added or subtracted. The distance of the protrusion or cutout can also be specified by entering a value in the **Distance** field on the Tool bar or floating data bar. The new solid is previewed on the drawing.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



4. Click at the desired location to edit the solid.

Direct Modeling with the Push/Pull Tool
Moving the Face of a Solid
Curve-based Addition or Subtraction

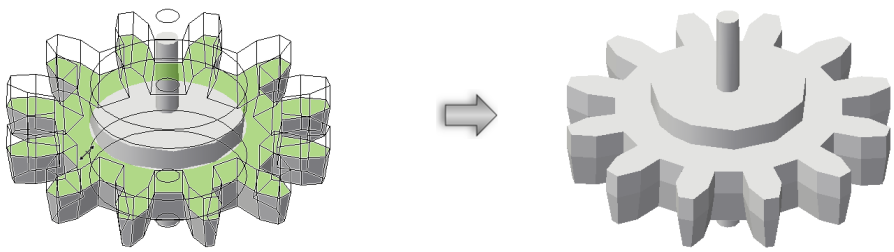
Moving the Face of a Solid

The **Move Face** mode extends the face of a solid, extruding the surrounding or adjacent geometry as the face moves.

To move the face of a solid in **Move Face** mode:

1. Click the **Push/Pull** tool from the 3D Modeling tool set, and then select **Move Face** from the Tool bar.
2. Click on the desired planar face.

   If needed, the Select Face dialog box opens to help select the correct face, as described in "Selecting the Edges and Faces of a Solid" on page 324. To select the back faces of solids, press the Alt key (Windows) or Option key (Mac) while selecting. When the correct face is selected, click **OK**.
3. The cursor changes to a double-headed arrow. Move the cursor to extend the face and its geometry. The distance can also be specified by entering a value in the **Distance** field on the Tool bar or floating data bar. The new solid is previewed on the drawing.
4. Click at the desired location to edit the solid.



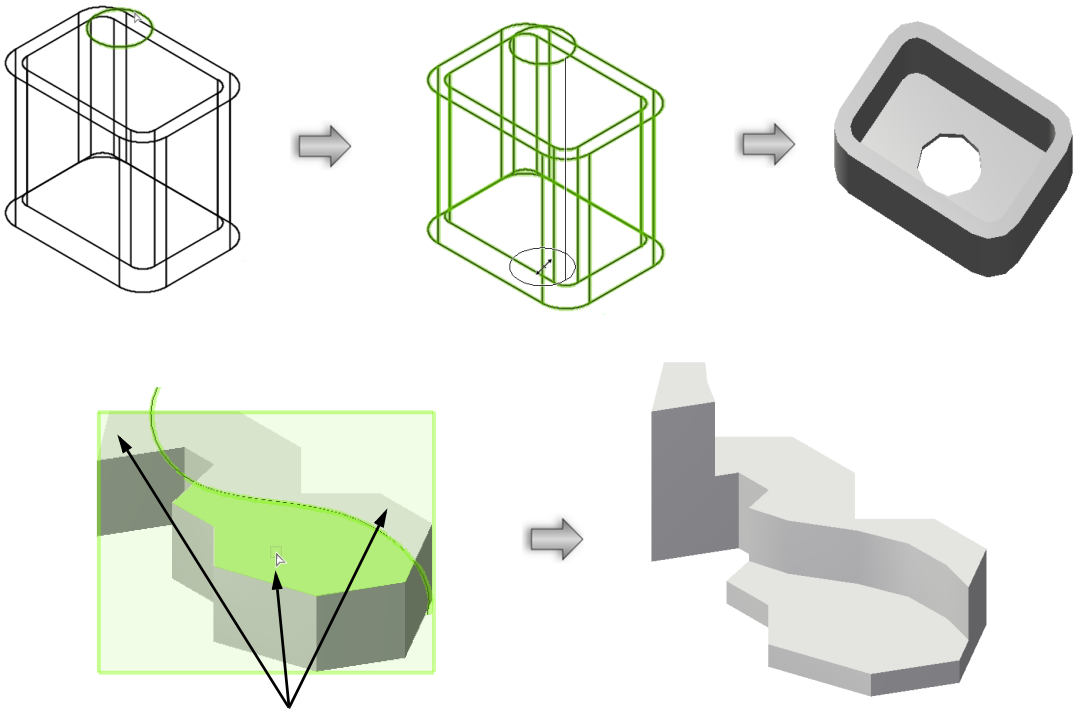
Direct Modeling with the Push/Pull Tool
Face-based Addition or Subtraction
Curve-based Addition or Subtraction

Curve-based Addition or Subtraction

To add or subtract from a solid in **Sub-Face** mode:

1. Click the **Push/Pull** tool from the 3D Modeling tool set, and then select **Sub-Face** from the Tool bar.
2. Select the NURBS curve or group of NURBS curves that will be used to create the protrusion or cutout, and then select the solid.
3. The cursor changes to a double-headed arrow. When an open NURBS curve is coplanar with the solid face and also splits the face of the solid, each split section can be moved individually. Move the cursor to expand (**Add** mode) or decrease (**Subtract** mode) the solid volume. The cursor direction determines whether volume is added or subtracted. The distance of the protrusion or cutout can also be specified by entering a value in the **Distance** field on the Tool bar or floating data bar. The new solid is previewed on the drawing.

   In **Subtract** mode, the distance value can exceed the solid perimeter; the cutout operation is only performed on the selected solid.



Each coplanar face can be moved individually

4. Click at the desired location to edit the solid.

Direct Modeling with the Push/Pull Tool
Face-based Addition or Subtraction
Moving the Face of a Solid

Tapering Faces

The **Taper Face** tool tapers faces of solid objects to a specified angle, with respect to a selected reference plane. This allows you to quickly and easily explore different solid shapes.

Two modes are available.

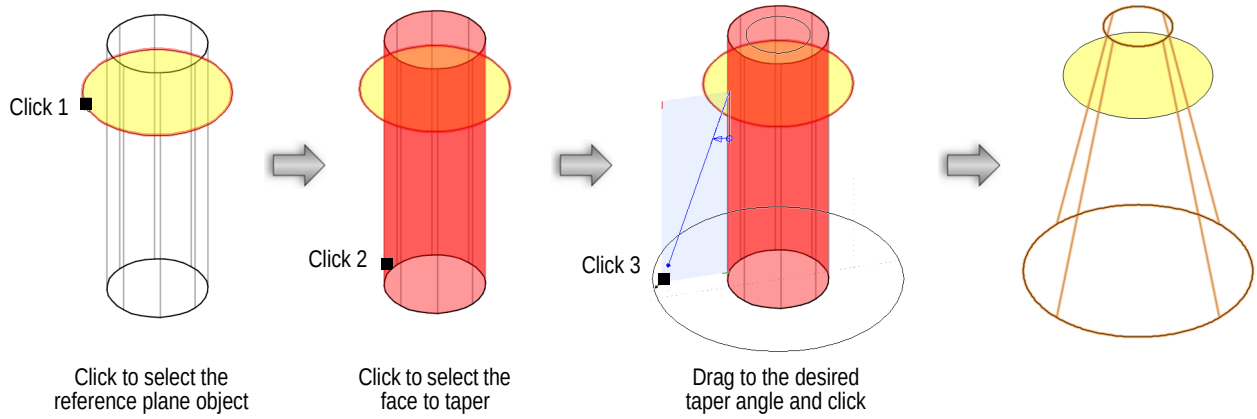
Mode	Description
Tangent Faces	Tapers the picked face and any faces that are tangent to it, as a group; for solids with curved sides
Picked Face	Tapers only the picked face

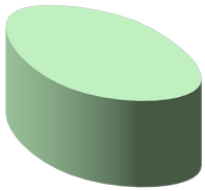
To taper a face of a solid:

1. Click the **Taper Face** tool from the 3D Modeling tool set, and then select a mode from the Tool bar.
2. Click to select an object or face of an object that will be used to determine the pivot point for the taper operation. This can be a 2D object such as a rectangle, the planar face of a solid (including the solid being tapered), or a planar NURBS curve or solid edge. Valid objects are highlighted when the cursor tracks over them.

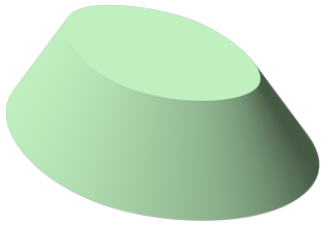
   Press the Alt (Windows) or Option (Mac) key to select an object or face that is behind another object.
3. Next, move the cursor to highlight the face to taper, and click to select it. In **Tangent Faces** mode, both the clicked face and the faces tangent to it are selected.

   Press the Alt (Windows) or Option (Mac) key to select a back face of an object.
4. Drag to display a preview of the taper. Alternatively, to taper to a precise angle, press the Tab key and enter a negative or positive number in the Data bar; press Enter (Windows) or Return (Mac) to display a preview of the taper.
5. Click, or press Enter to complete the taper operation. The result is a generic solid object.

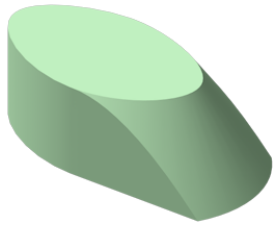




The original object is an extruded NURBS curve



In **Tangent Faces** mode, all of the side faces are tangent to each other, so they are tapered as a group



In **Picked Faces** mode, only the front face is selected and tapered

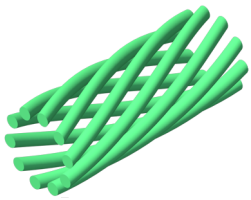
Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

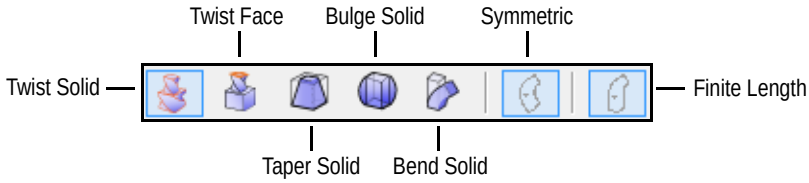
Tapering a Solid or NURBS Surface

Deforming Solids and NURBS Surfaces

The **Deform** tool reshapes solid objects and NURBS surfaces to allow for easy exploration of sculptural shapes. The tool twists, tapers, bulges, and bends existing solid objects; the resulting objects are generic solids.



Five primary modes are available. Two additional modes are activated depending on the primary mode selected.



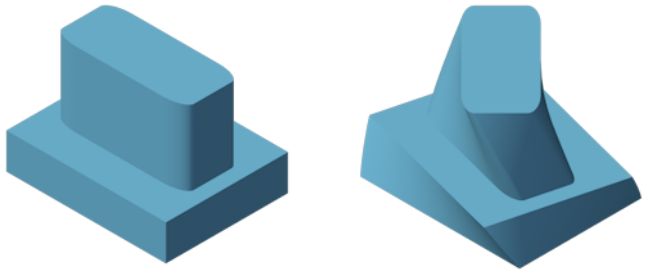
Mode	Description
Twist Solid	Twists the selected solid or NURBS surface to a specified angle
Twist Face	Twists the selected face and the adjacent and tangent faces of a solid to a specified angle
Taper Solid	Tapers the selected solid or NURBS surface to a specified ratio
Bulge Solid	Bulges the selected solid or NURBS surface according to a specified ratio
Bend Solid	Bends the selected solid or NURBS surface to a specified angle
Symmetric	Deforms the solid symmetrically
Finite Length	Deforms the solid within the length specified by the bend spine; the portion of the shape beyond the bend spine maintains the original tangency

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Twisting an Entire Solid or NURBS Surface
Twisting the Faces of a Solid
Tapering a Solid or NURBS Surface
Creating a Bulged Solid or NURBS Surface

Bending a Solid or NURBS Surface

Twisting an Entire Solid or NURBS Surface



In **Twist Solid** mode, the entire solid object is twisted

To twist a solid or NURBS surface:

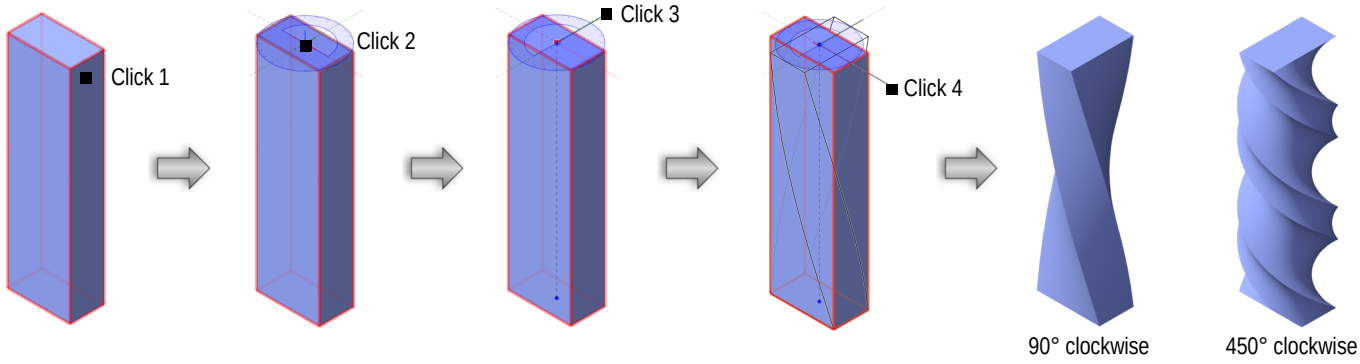
1. Click the **Deform** tool from the 3D Modeling tool set, and then select **Twist Solid** mode.
2. As you move the mouse over 3D objects, they are highlighted to indicate they can be modified. Click to select the solid or NURBS surface.

   Shift-click to select more than one object to twist.
3. A protractor feedback graphic displays around the cursor. Position the protractor on the appropriate twist plane, and click at the center point of the twist.

   The protractor graphic displays only when an object is selected. To position the protractor as desired, you may need to activate a specific plane using the Active Planes list on the View bar.

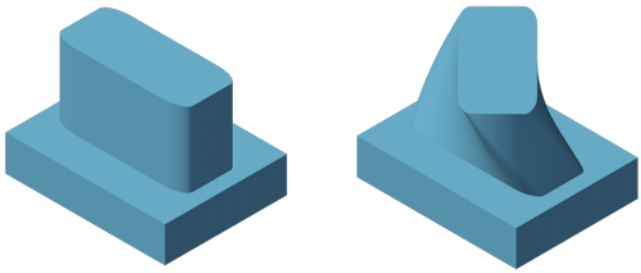
   The center point of the twist can be located away from the object being twisted.
4. The protractor graphic remains at the click point, and the twist axis line appears. As you move the cursor, the protractor rotates, and the twist reference line appears and previews the starting point for the twist. Click to place the reference line.
5. As you move the cursor, the twist angle line tracks the cursor, and a preview of the twisted object displays.

   Objects can be twisted more than 360 degrees.
6. Click to complete the operation. The result is a generic solid object.



Twisting the Faces of a Solid
Deforming Solids and NURBS Surfaces

Twisting the Faces of a Solid

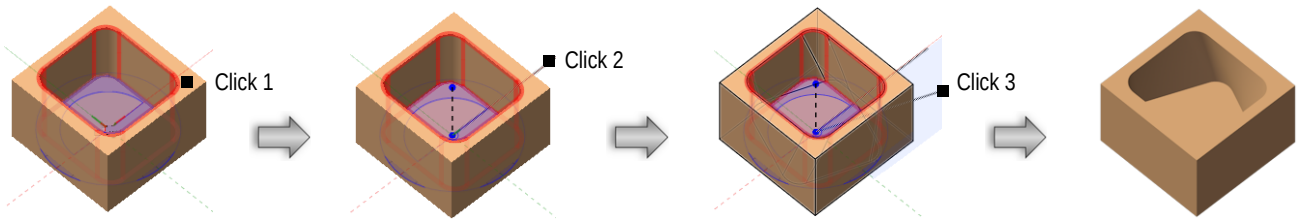


In Twist Face mode, only the selected face and the adjacent and tangent faces are twisted

To twist the faces of a solid:

1. Click the **Deform** tool from the 3D Modeling tool set, and then select **Twist Face** mode. A protractor feedback graphic displays around the cursor.
2. As you move the protractor over 3D objects, the faces to be twisted are highlighted. Position the protractor on the appropriate face, and click at the center point of the twist.

   The protractor is oriented according to the face normal. Face mode twists the selected face and all adjacent and tangent faces as indicated by the highlight.
3. The protractor graphic remains at the click point, and the twist axis line appears. As you move the cursor, the protractor rotates, and the twist reference line appears and previews the starting point for the twist. Click to place the reference line.
4. As you move the cursor, the twist angle line tracks the cursor, and a preview of the twisted object displays.
5. Click to complete the operation. The result is a generic solid object.



Twisting an Entire Solid or NURBS Surface
Deforming Solids and NURBS Surfaces

Tapering a Solid or NURBS Surface

To taper a solid or NURBS surface:

1. Click the **Deform** tool from the 3D Modeling tool set, and then select **Taper Solid** mode.

   Symmetric mode is selected by default; a four-way arrow feedback graphic displays around the cursor, and an equal amount of taper applies to both the drag direction and its perpendicular direction. If an asymmetrical taper is desired, deselect **Symmetric** mode on the tool bar; a two-way arrow feedback graphic displays, and the taper applies only to the drag direction; its perpendicular direction has no taper.

   The **Symmetric** mode status can also be changed at any time during the operation.

2. As you move the mouse over 3D objects, they are highlighted to indicate they can be modified. Click to select the solid or NURBS surface.

   Shift-click to select more than one object.

3. A four-way or two-way arrow feedback graphic displays around the cursor. Position the arrow graphic at the desired taper center, and click.

   The arrow graphic displays only when an object is selected. To position the arrows as desired, you may need to activate a specific plane using the Active Planes list on the View bar.

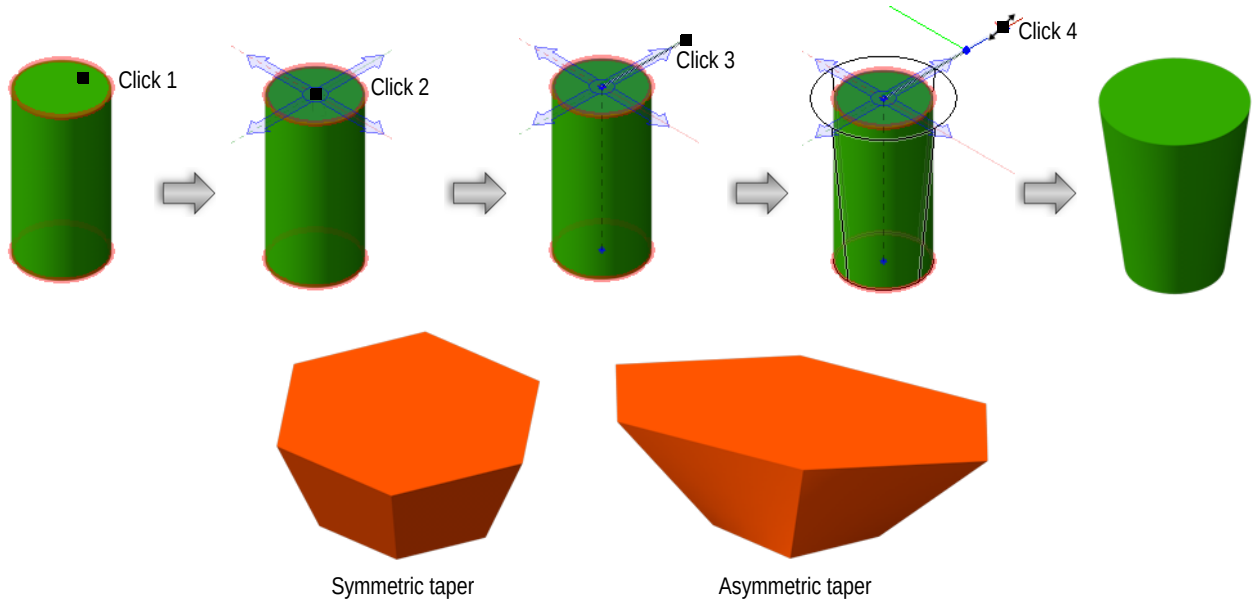
   The taper center can be located away from the object being tapered.

4. The arrow graphic remains at the click point, and the taper axis line appears. As you move the cursor, the arrows rotate to indicate the direction of the taper, and the taper reference line previews the starting point for the taper. Click to place the reference line.

5. As you move the cursor, the taper ratio line tracks the cursor, and a preview of the tapered object displays.

   To taper to a precise ratio, press the Tab key and enter a positive number in the Data bar. The original shape ratio is 1; numbers below 1 taper inward, while numbers above 1 taper outward. Press Enter to display a preview of the taper.

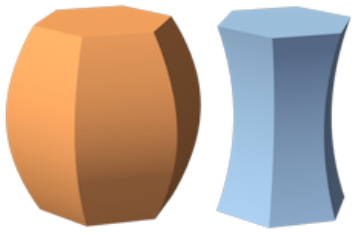
6. Click to complete the operation. The result is a generic solid object.



Tapering Faces
Deforming Solids and NURBS Surfaces

Creating a Bulged Solid or NURBS Surface

The **Deform** tool's **Bulge Solid** mode can be used to create a convex or concave shape.



To create a bulge in a solid or NURBS surface:

1. Click the **Deform** tool from the 3D Modeling tool set, and then select **Bulge Solid** mode.

   Symmetric mode is selected by default; a four-way arrow feedback graphic displays around the cursor, and an equal amount of bulge applies to both the drag direction and its perpendicular direction. If an asymmetrical bulge is desired, deselect **Symmetric** mode on the tool bar; a two-way arrow feedback graphic displays, and the bulge applies only to the drag direction; its perpendicular direction has no bulge.

   The **Symmetric** mode status can also be changed at any time during the operation.

2. As you move the mouse over 3D objects, they are highlighted to indicate they can be modified. Click to select the solid or NURBS surface.

   Shift-click to select more than one object.

3. A four-way or two-way arrow feedback graphic displays around the cursor. Position the arrow graphic at the desired bulge center and click.

   The arrow graphic displays only when an object is selected. To position the arrows as desired, you may need to activate a specific plane using the Active Planes list on the View bar.

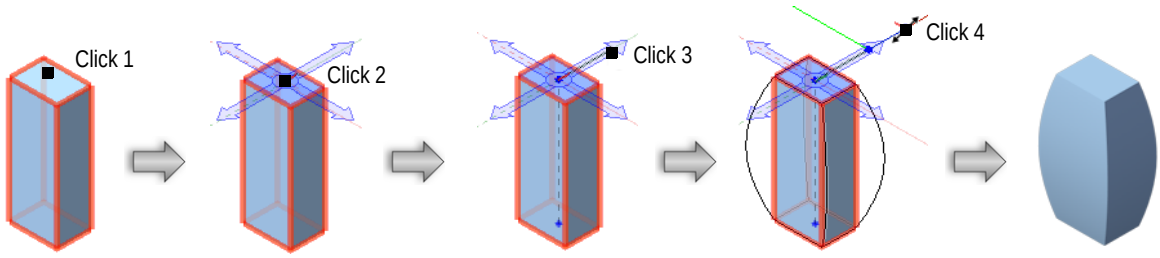
   The bulge center can be located away from the object being bulged.

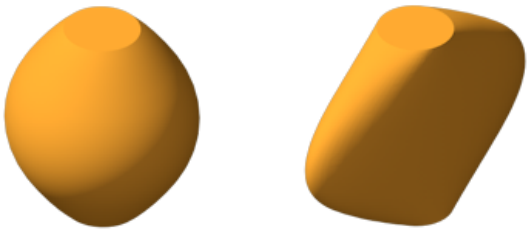
4. The arrow graphic remains at the click point, and the bulge axis line appears. As you move the cursor, the arrows rotate to indicate the direction of the bulge, and the bulge reference line previews the starting point for the bulge. Click to place the reference line.

5. As you move the cursor, the bulge ratio line tracks the cursor, and a preview of the bulged object displays.

   To create the bulge to a precise ratio, press the Tab key and enter a positive number in the Data bar. The original shape ratio is 1; numbers below 1 bulge inward, while numbers above 1 bulge outward. Press Enter to display a preview of the bulge.

6. Click to complete the operation. The result is a generic solid object.





Symmetric bulge Asymmetric bulge

Deforming Solids and NURBS Surfaces

Bending a Solid or NURBS Surface

To bend a solid or NURBS surface:

1. Click the **Deform** tool from the 3D Modeling tool set, and then select **Bend Solid** mode.

   Symmetric mode is selected by default; the bend applies to both the positive and negative direction of the bend spine. If an asymmetrical bend is desired, deselect **Symmetric** mode on the tool bar; the bend applies only to the positive direction of the bend spine.

   Finite Length mode is selected by default; the bend applies only to the finite length of the bend spine. To bend the entire object, deselect **Finite Length** mode on the tool bar.

   The **Symmetric** mode and **Finite Length** mode status can also be changed at any time during the operation.

2. As you move the mouse over 3D objects, they are highlighted to indicate they can be modified. Click to select the solid or NURBS surface.

   Shift-click to select more than one object.

3. A protractor feedback graphic displays around the cursor. Position the protractor on the appropriate bend spine starting point, and click.

   The protractor graphic displays only when an object is selected. To position the protractor as desired, you may need to activate a specific plane using the Active Planes list on the View bar.

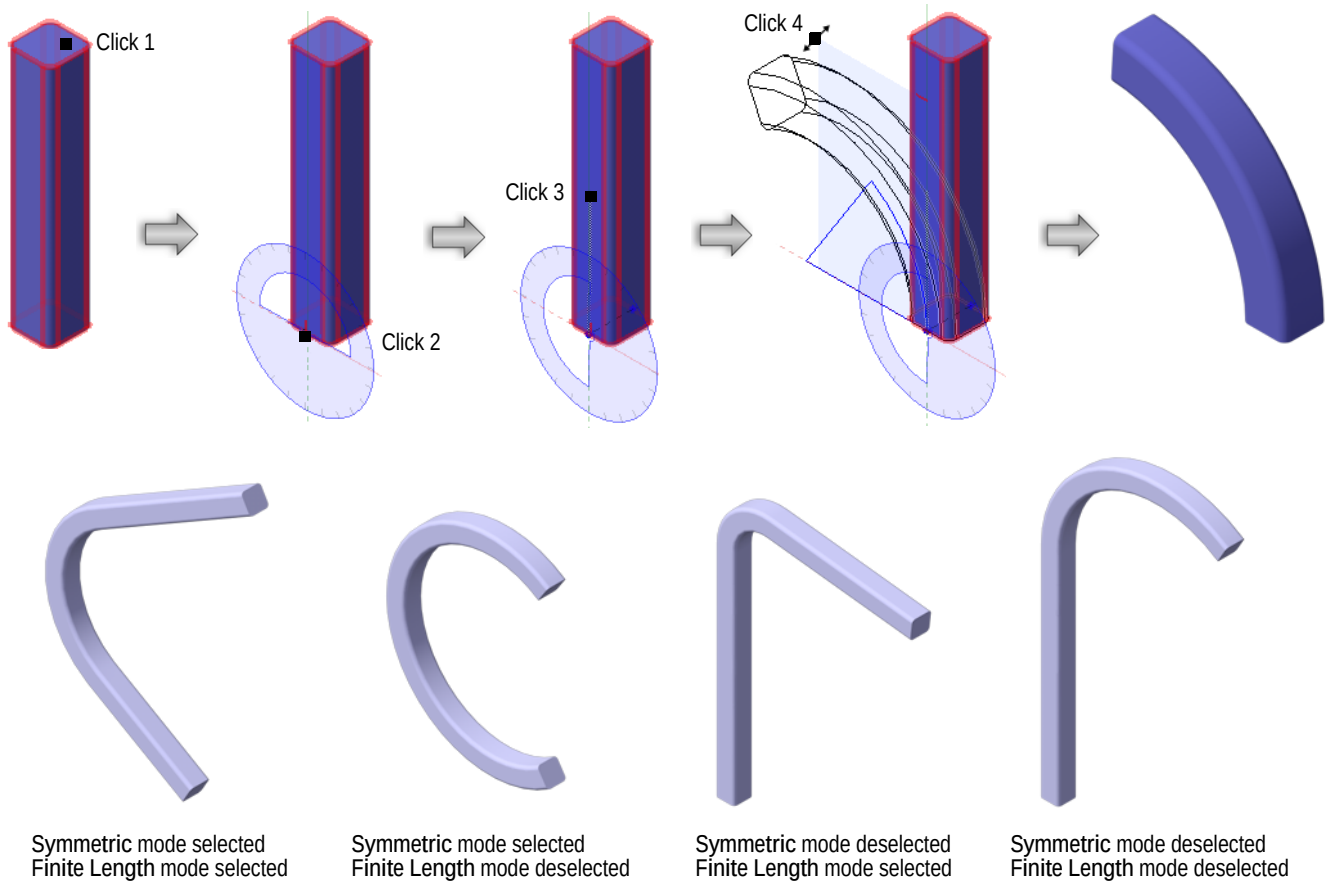
4. The protractor remains at the click point, and the protractor normal line appears. As you move the cursor, the protractor rotates and the bend spine appears; the line previews the bend spine's end point. Click to place the bend spine.

   If **Finite Length** mode is selected, the bend spine, which is defined by click 2 and click 3, delimits the section to bend.

5. As you move the cursor, the bend spine tracks the cursor, and a preview of the bent object displays.

To bend to a precise angle, press the Tab key and enter a negative or positive number in the Data bar; press Enter to display a preview of the bend.

6. Click to complete the operation. The result is a generic solid object.



Deforming Solids and NURBS Surfaces

Chamfering Solid Edges

The **Chamfer Edge** tool modifies the edge of a solid with a linear cross section. Edges are usually modified during the detailed design phase of a project to provide a more realistic or aesthetic appearance to a model.

To chamfer the edges of a solid:

1. Click the **Chamfer Edge** tool from the 3D Modeling tool set, and then select **Preferences** on the Tool bar.

   The Chamfer Edge Preferences dialog box opens.

Click to show/hide the parameters.

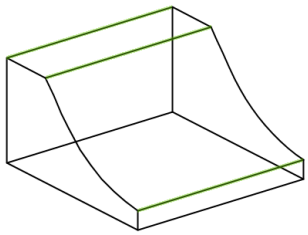
Parameter	Description
Setback	Specify the distance by which the faces are set back Alternatively, enter a value in the **Setback** field on the Tool bar
Select Tangent Entities	Choose this option to select tangentially connected edges and faces for chamfering

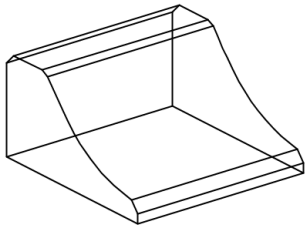
Parameter	Description
Select Faces	Choose this option to select faces for chamfering
Select All Edges	Selects all the edges of the solid for the chamfer operation

2. Enter your chamfer preferences, and click **OK**.
3. For tangent entity selections, select the tangentially connected edges and faces to be modified. For face selections, select the faces to be modified. To select multiple edges or surfaces, hold the Shift key while selecting. To select the back faces of solids, press the Alt key (Windows) or Option key (Mac).

   See “Selecting the Edges and Faces of a Solid” on page 324 for information on selecting surfaces.

   When **Select All Edges** is enabled, clicking on one of the highlighted edges selects all the edges for chamfering.
4. Press Enter or click the check mark button on the Tool bar to perform the edge modification.


Before chamfering


After chamfering


Results rendered

The **Setback** value for the chamfered edge can be edited through the Object Info palette.

Once the modification is complete, the edge set used to create the modification cannot be changed. The modified edge can be ungrouped, and a new modification with different edges can be created.

If one modification has been created over another, the first modification cannot be edited in the Object Info palette unless the last one is ungrouped (see “Editing a Fillet/Chamfer or Shell Object” on page 327).

A chamfer edge operation may fail due to any of the following:

- Complex surface geometry adjacent to an edge or corner involved in the modification
- An attempt to modify one edge in a sequence of tangential edges (select all the edges for a better chance of success)
- An attempt to modify without selecting all the edges at a vertex

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Filleting Solid Edges
NURBS Curves and Surfaces

Filleting Solid Edges

The **Fillet Edge** tool modifies the edge of a solid with a circular cross section. Edges are usually modified during the detailed design phase of a project to provide a more realistic or aesthetic appearance to a model.


To fillet the edges of a solid:

1. Click the **Fillet Edge** tool from the 3D Modeling tool set, and then select **Preferences** on the Tool bar.

   The Fillet Edge Preferences dialog box opens.

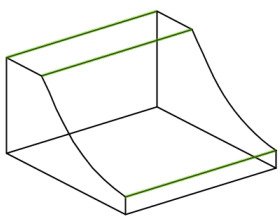
Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Select Tangent Entities	Choose this option to select tangentially connected edges and faces for filleting
Constant Radius	Create a filleted surface based on a constant radius
Radius	Indicate the radius of the fillet edge Alternatively, enter a value in the **Constant Radius** field on the Tool bar
Select Faces	Choose this option to select faces for filleting
Select All Edges	Selects all the edges of the solid for the fillet operation
Variable Radius	Create a filleted surface based on a variable radius
Length (%)	Enter the length up to the point as a percentage of the total edge length (for example, 50)
Radius	Enter the radius value at the corresponding percentage of length value (for example, 1”)
Set	Click to add a length/radius pair or modify the currently selected pair
Delete	Click to delete the selected length/radius value
Reverse	Reverses the order of the radius values; the radius values that were at the shortest percentage length are switched to the longest percentage length, and vice versa

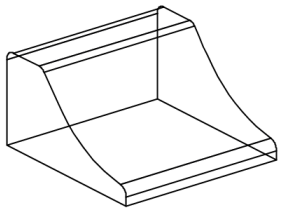
2. Enter your fillet preferences, and click **OK**.
3. For tangent entity selections, select the tangentially connected edges and faces to be modified. For face selections, select the faces to be modified. To select multiple edges or surfaces, hold the Shift key while selecting. To select the back faces of solids, press the Alt key (Windows) or Option key (Mac).

   See “Selecting the Edges and Faces of a Solid” on page 324 for information on selecting surfaces.

   When **Select All Edges** is enabled, clicking on one of the highlighted edges selects all the edges for filleting.
4. Press Enter or click the check mark button on the Tool bar to perform the edge modification.

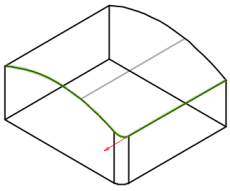


Before filleting

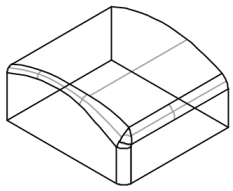


After filleting with a constant radius

Results rendered



Before filleting



After filleting with a variable radius

Results rendered

The **Radius** value for the filleted edge can be edited through the Object Info palette. For the fillet by variable radius, press the **Edit** arrow buttons to highlight each point along the edge for editing.

Once the modification is complete, the edge set used to create the modification cannot be changed. The modified edge can be ungrouped, and a new modification with different edges can be created.

If one modification has been created over another, the first modification cannot be edited in the Object Info palette unless both are first ungrouped (see "Editing a Fillet/Chamfer or Shell Object" on page 327).

A fillet edge operation may fail due to any of the following:

- Complex surface geometry adjacent to an edge or corner involved in the modification
- A radius value that is too large
- An attempt to modify one edge in a sequence of tangential edges (select all the edges for a better chance of success)
- An attempt to modify without selecting all the edges in a vertex

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Chamfering Solid Edges
NURBS Curves and Surfaces

Creating a Shell from Solids, NURBS Surfaces, and Planar Objects

The **Shell Solid** tool creates a hollow shell from a solid object or gives thickness to NURBS surfaces and planar objects. Most injection-molded plastic parts have a shell.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Shell from a Solid Object
Shell from a NURBS Surface
Shell from a Planar Object
NURBS Curves and Surfaces

Shell from a Solid Object

To create a shell solid from a solid object:

1. Click **Shell Solid** from the 3D Modeling tool set, and then select **Preferences** on the Tool bar.

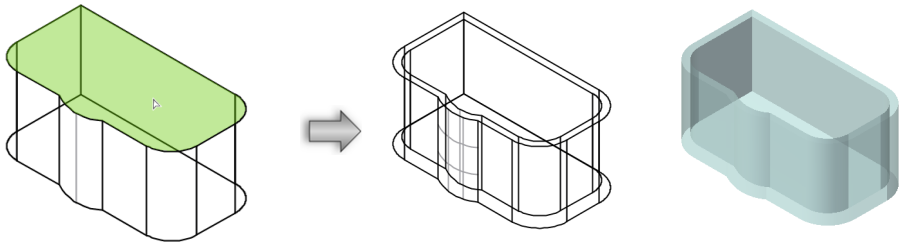
   The Shell Solid Preferences dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Shell	
Inside	Creates an interior shell
Outside	Creates an exterior shell
Thickness	Enter the thickness of the shell wall Alternatively, enter a value in the **Thickness** field on the Tool bar
Select Tangent Faces	Selects a tangentially connected chain of faces

2. Enter your shell preferences, and click **OK**.
3. Click on the face of the object where the shell will be created. To select multiple surfaces, hold the Shift key while selecting. To select the back faces of solids, press the Alt key (Windows) or Option key (Mac).

   See "Selecting the Edges and Faces of a Solid" on page 324 for information on selecting surfaces.
4. Press Enter or click the check mark button on the Tool bar to create the shell.



Click here for a video tip on this topic (Internet connection required).

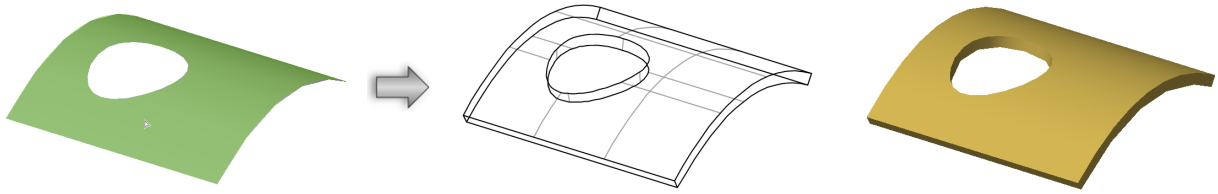
Shell from a NURBS Surface
Shell from a Planar Object
Editing a Fillet/Chamfer or Shell Object
NURBS Curves and Surfaces

Shell from a NURBS Surface

To create a shell solid from a NURBS surface:

1. Click **Shell Solid** from the 3D Modeling tool set, and then select **Preferences** on the Tool bar.
2. Enter your shell preferences as described in "Shell from a Solid Object" on page 370, and click **OK**.

   For NURBS surfaces, the Inside setting extends the shell solid in the opposite direction from the surface's normal, and the Outside setting extends the shell solid in the direction of the surface's normal.
3. Click on the NURBS surface to thicken. To select multiple surfaces, hold the Shift key while selecting.
4. Press Enter or click the check mark button on the Tool bar to create the shell.



Shell from a Planar Object
Editing a Fillet/Chamfer or Shell Object
NURBS Curves and Surfaces
Displaying Surface Normals

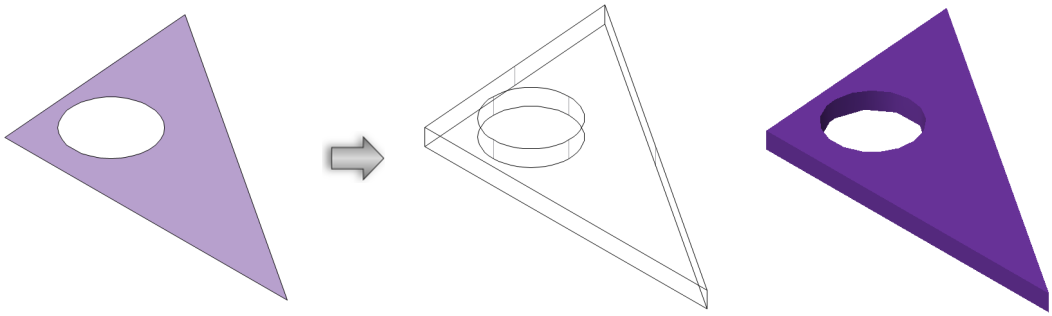
Shell from a Planar Object

To create a shell solid from a planar object:

1. Click **Shell Solid** from the 3D Modeling tool set, and then select **Preferences** on the Tool bar.
2. Enter your shell preferences as described in "Shell from a Solid Object" on page 370, and click **OK**.

   For planar objects, the **Inside/Outside** setting determines on which side of the plane the shell solid will be extended; the direction depends on several factors related to the object's creation, geometry, and placement in the document.
3. Click on the planar object to thicken. To select multiple objects, hold the Shift key while selecting.

4. Press Enter or click the check mark button on the Tool bar to create the shell.



Editing a Fillet/Chamfer or Shell Object
Shell from a NURBS Surface

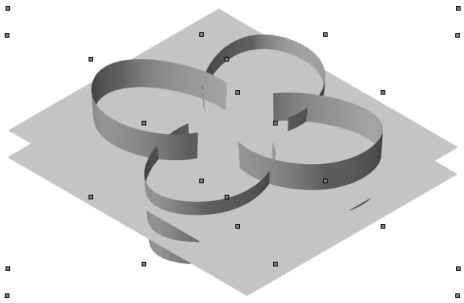
Stitching and Trimming Surfaces

The **Stitch and Trim Surfaces** command creates a generic solid from several intersecting surfaces (including NURBS surfaces or solids such as extrudes or sweeps).

> This command may not be able to manipulate certain types of surface geometry. See “Surface Geometry Requirements” on page 327.

To stitch and trim intersecting surfaces:

1. Select the intersecting surfaces.



> The surfaces must enclose a volume.

2. Select **Model > 3D Power Pack > Stitch and Trim Surfaces**.

   The surfaces enclosing the volume are stitched together and then trimmed, resulting in a generic solid.

Selecting the Edges and Faces of a Solid
NURBS Curves and Surfaces

Converting to Generic Solids

Solid objects created using such commands as **Add Solid**, **Subtract Solid**, and tools such as **Fillet Edge**, **Chamfer Edge**, and **Shell Solid**, contain a history composed of the original elements used to make the new object. This also includes any editing performed with other solids operations. The **Convert to Generic Solids** command removes this history from the solid objects, reducing the file size (though the object is no longer editable).

For example, use this command on a copy of the final drawing to reduce file size when sending a file to a print bureau.

To convert an object into a generic solid:

1. Select the solid object(s) to convert.
2. Select **Modify > Convert > Convert to Generic Solids**.
3. A warning dialog box opens. Click **OK** to convert the object(s).

   The object is converted.

   If objects are selected that cannot be converted, a dialog box opens stating that these objects have been deselected.

D Unfolding Surfaces

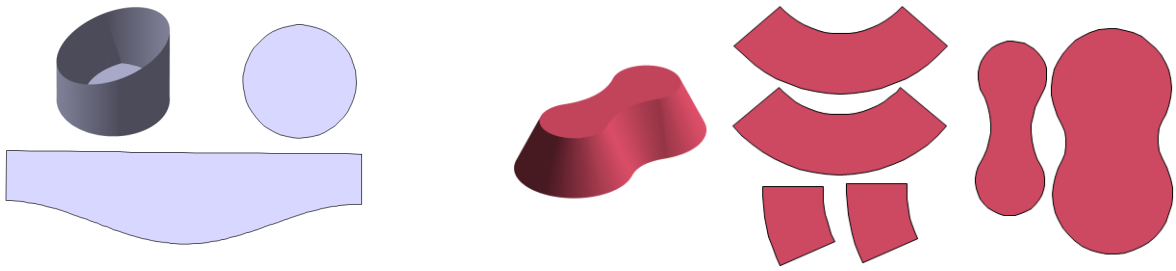
The **Unfold Surfaces** command creates flattened, 2D representations of developable surfaces. A developable surface is curved in one dimension only, so that it is able to be flattened into a plane with no distortion. For example, cones and cylinders can be unfolded, but spheres cannot.

To unfold surfaces:

1. Select one or more developable NURBS surfaces or CSG objects that represent developable surfaces.
2. Select **Model > 3D Power Pack > Unfold Surfaces**.

   A polyline in the shape of each unfolded surface is created and placed in the active layer. Each polyline has the same class and attributes as the surface from which it was created.

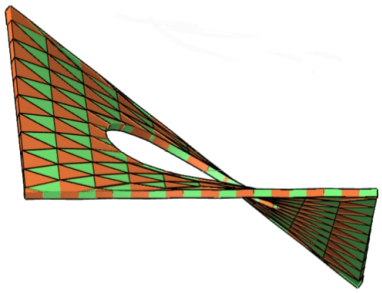
   The **Unfold Surfaces** command works on a single NURBS surface or a solid containing tangentially-connected NURBS surfaces. If the item does not successfully unfold, use the **Extract** tool to extract a NURBS surface from the item, and unfold that surface.

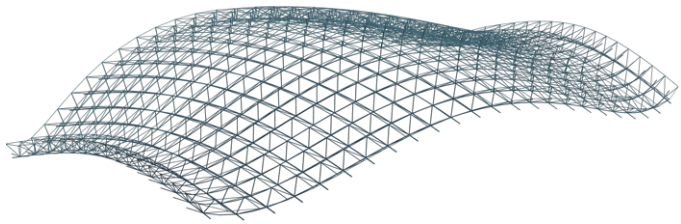


Click here for a video tip about this topic (Internet access required).

D Creating a Surface Array

The **Create Surface Array** command duplicates 2D or 3D geometry onto a planar object or NURBS surface. This allows you to easily create free-form surfaces with geometric patterns, curtain walls, open framework, and more.





The following base surface objects are allowed.

- Arc
- Circle
- Oval
- Polygon
- Polyline
- Rectangle
- Rounded rectangle
- NURBS surface

The following table describes the types of objects that can be used as array items, as well as the method used to project the objects onto the base surface.

Array Item	Projection Method
Solid Sweep Text	Project by center of bounding cube
Symbol instance	Project by symbol origin
2D/3D Locus Arc Bitmap Line Oval Polygon Polyline Rectangle Rounded rectangle	Project by object definition points (the result is always a planar object)
Extrude/multi-extrude	Project profile object by definition points
3D Poly Extrude along path Mesh NURBS curve NURBS surface	Project by each vertex (the result may not be planar)
Group	Project member objects as described above; all members of the group must be legal array objects

To create a surface array:

1. Select a planar object or NURBS surface to be the base of the array object. Optionally, also select the 2D or 3D geometry that will be duplicated on that surface (alternatively, edit the surface array object after creation to add the array geometry).

2. Select **Model > Create Surface Array**.

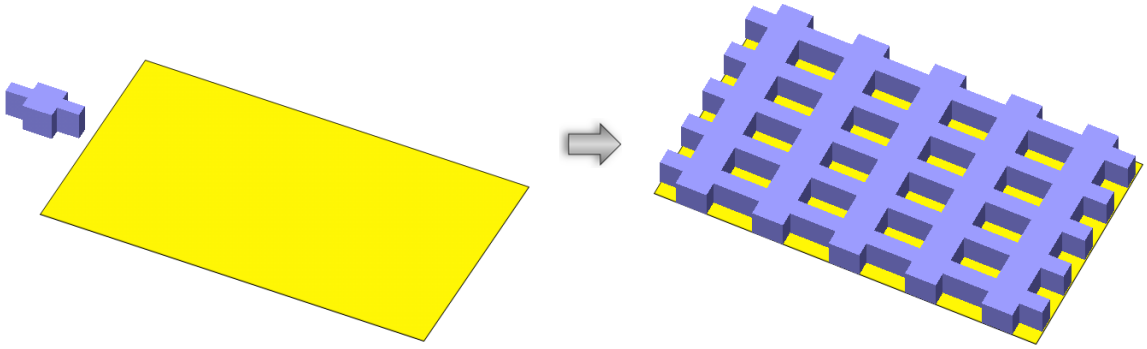
   The Create Surface Array dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Select a base surface object	If multiple objects were selected, by default, the largest object is highlighted in red, which indicates that it is selected to be used as the base surface. Click the **Next** or **Prev** button to highlight a different object. The buttons are unavailable if only one object was selected.
Origin X' Factor	Enter the X offset of the array pattern's origin from the center of the base surface. The value must be a number between -1 and 1. For example, to shift each array item to the right by half of the item's length along the X axis, enter 0.5.
Origin Y' Factor	Enter the Y offset of the array pattern's origin from the center of the base surface. The value must be a number between -1 and 1. For example, to shift each array item down by a quarter of the item's length along the Y axis, enter -0.25.
Pattern Rotation	Enter a rotation angle for the array pattern
X and Y Repetition Parameters	For both the X and Y axes, specify how the array objects are to be repeated
Repetition Mode X' / Y'	Select whether to repeat array objects on the base surface based on a **No. of Repetitions**, or a **Fixed Distance** per object
No. of Repetitions X' / Y'	If **Repetition Mode** is set to Number of Repetitions, enter the number of times the array object is to be repeated along the X or Y axis. The array objects are resized as necessary to fit the base surface.
Fixed Distance X' / Y'	If **Repetition Mode** is set to Fixed Distance, enter the length of each array object along the X or Y axis. The number of array objects is adjusted as necessary to fit the base surface.
Offset X' / Y' Factor	Enter a number to control the space between the array objects. The distance from the center of one array item to the center of the next item (in the same row or column) is the factor times the array item's size. For example, if the array item is 12" total, and the **Offset X' Factor** is 1.25, the distance from the center of one item to the next in the same row is 15" (12 x 1.25). If the offset factor is 1, the array items in that row or column touch each other; if the factor is negative, the array items overlap.
Shift X' / Y' Factor	Enter a number to shift the position of the array objects from one row or column to the next. The distance from the center of one array item to the center of an array item in the next row or column is the factor times the array item's size. For example, if the array item is 12" total, and the **Shift Y' Factor** is 0.25, the distance from the center of one item to the center of the item to the right (in the next column) is 3" (12 x 0.25). If the shift factor is 0 (zero), the array items are aligned.
Edge Conditions	Specify how to handle array objects that extend beyond the boundary of the base surface: allow the objects to Overlap the edge, Omit those objects from the array, or Trim the objects at the boundary
Display Base Surface	Select whether to display or hide the base surface

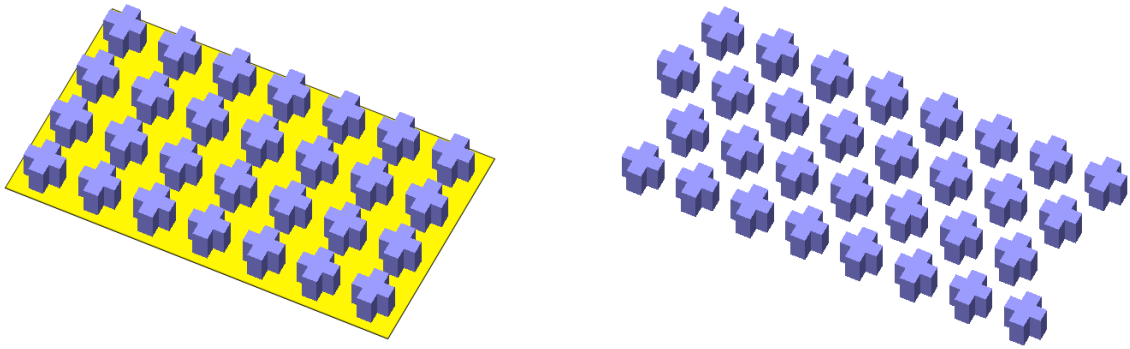
Parameter	Description
Scale Z' Value	When selected, the array objects are automatically scaled along the Z axis, proportional to how much the objects are scaled along the X and Y axes (to fit the base surface). When deselected, the array objects have the same Z value as the original array item.

3. Enter the surface array parameters, and click **OK** to create the object.



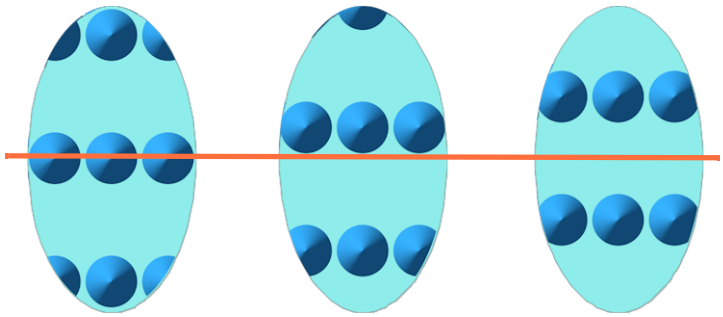
The base surface is a yellow rectangle, and the array item is a blue extrude

Surface array with 5 repetitions along both the X and Y axes. Offset factors are set to 1, so there are no gaps between the array items. Shift factors are set to 0, so the array items are aligned.



No. of Repetitions set to X=7 and Y=4, and offset factor set to 1.5 for both X and Y axes

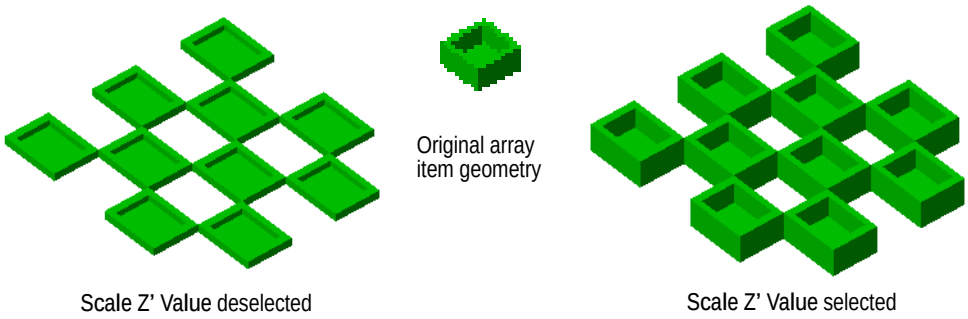
Shift X' Factor set to 0.5, and Display Base Surface deselected



Origin Y' Factor = 0; the array is centered at the base surface origin

Origin Y' Factor = 0.5; the center of the array is offset above the origin by half the width of an array item

Origin Y' Factor = 1; the center of the array is offset above the origin by the full width of an array item



Click here for a video tip about this topic (Internet access required).

Editing a Surface Array

After a surface array has been created, you can edit its properties from the Object Info palette. All of the parameters that are available during the array creation can be changed. You can also change which side of the base surface the array is applied to (**Switch Z' Direction**).

You cannot edit the surface array geometry directly; use the **Edit Surface Array** command to edit the base surface and array components individually.

To edit a base surface or array item:

1. Select the surface array, and then select **Modify > Edit Surface Array**.
2. From the Choose component dialog box, select whether to edit the base surface or the array items.

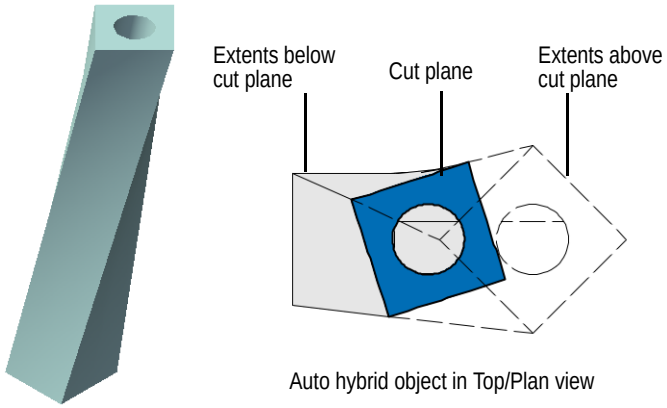
   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the surface array and select **Edit Base Surface** or **Edit Array Items** from the context menu.
3. The Edit window displays the item to be edited. A colored border around the drawing window indicates the editing mode is active.
4. Reshape the geometry, change the attributes, and move, add, or remove items as needed. Keep in mind that only one item can be used as the base surface. Also, the base surface and array item(s) must be one of the allowed objects types described previously.
5. Click the **Exit** button in the top right corner of the drawing window to return to regular drawing mode and update the surface array.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Object Editing Mode

D Creating Auto Hybrid Objects

Converting 3D geometry into an auto hybrid object allows the object to appear as desired in 2D plan view, while leaving the 3D appearance unchanged. This is particularly important for Building Information Model (BIM) workflows and for those who model free-form 3D geometry, but need plan drawings as well. The auto hybrid's settings provide complete control over its 2D display attributes, with separate, classed parameters for the appearance of the cut plane and extents below and above the cut plane.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Auto hybrid object in Top/Plan view

To create an auto hybrid object:

1. Select the 3D geometry to convert to an auto hybrid object.

   Valid objects include: extrudes, tapered extrudes, multiple extrudes, extrude along path objects, 3D polygons, sweep objects, meshes, solids, NURBS surfaces, and 3D-only symbols and 3D plug-in objects.

2. If not in Top/Plan view, select **View > Standard Views > Top Plan**.

   The auto hybrid's 2D appearance settings are only visible in Top/Plan view.

3. With the object selected, select the **Create Auto Hybrid** command from the appropriate menu:
   - Architect workspace: **AEC > Create Auto Hybrid**
   - Landmark workspace: **Landmark > Architectural > Create Auto Hybrid**
   - Spotlight workspace: **Spotlight > Architectural > Create Auto Hybrid**

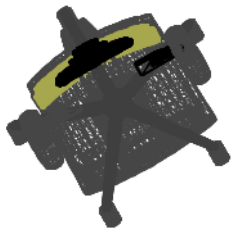
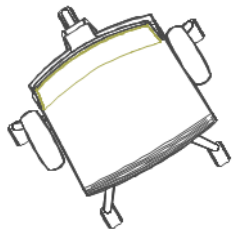
4. The object is converted into an auto hybrid.

5. To set the location of the cut plane and specify the appearance of the auto hybrid, click **2D Appearance** from the Object Info palette.

   The 2D Appearance dialog box opens. Click on the Cut Plane tab and set the desired parameters.

Click to show/hide the parameters.

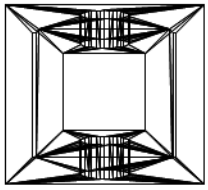
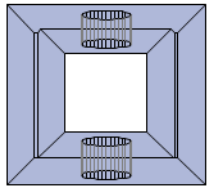
Parameter	Description
Elevation	Sets the elevation of the cut plane; the cut plane is the intersection of an imaginary XY plane and the object geometry. Specify whether the elevation is relative to the layer, or to the story (Vectorworks Architect required). The cut plane is set to this fixed elevation and does not move with the object; instead, if you move the object's elevation, the cut plane displays in a different location along the object. If the cut plane does not intersect the object, the cut plane does not display, but the object does.
Display Cut Plane	Select to display the cut plane of the auto hybrid object
Class	Select the class to use from the current list of classes, select New to create a new class, or select <Contained Object Class(es)> to place the cut plane in the same class(es) as the 3D contained object(s).
Fill	
None	Uses no fill for the cut plane
Use 2D Attributes of Contained Object(s)	Uses the fill attributes of the original converted object(s) for the cut plane

Parameter	Description
Use Class 2D Attributes	Uses the **Class** selection for the cut plane fill attributes
Pen	
Use 2D Attributes of Contained Object(s)	Uses the pen attributes of the original converted object(s) for the cut plane
Use Class 2D Attributes	Uses the **Class** selection for the cut plane pen attributes
Show as Reflected Ceiling Plan	When selected, displays all the auto hybrid geometry as if looking up, rather than looking down; applies to all tabs
3D Conversion Resolution	Sets the segmentation resolution used to display curved portions of the auto hybrid; this setting is independent of the **3D conversion resolution** Vectorworks preference and applies to all tabs
Smoothing Angle	When **Display Extents Below/Above Cut Plane** is selected on either the Below Cut Plane or Above Cut Plane tab, set a value to reduce the number of facets displayed in the geometry. For values greater than zero, facet lines are removed between any two faces of an object that are within that degree angle of each other; applies to all tabs.  Original Top/Plan view of office chair (comprising 90 meshes)  Office chair auto hybrid object with Smoothing Angle setting = 5°
Generate Intersecting Lines	When **Display Extents Below/Above Cut Plane** is selected on either the Below Cut Plane or Above Cut Plane tab, select to generate lines between intersecting geometry or deselect to omit intersecting lines; applies to all tabs. This option may make generating an auto hybrid object slower.
Save Current Settings as Defaults	Applies the current settings from all tabs as default parameters when creating auto hybrid objects

6. Click on the Below Cut Plane/Above Cut Plane tabs to set the appearance of the auto hybrid below/above the cut plane.

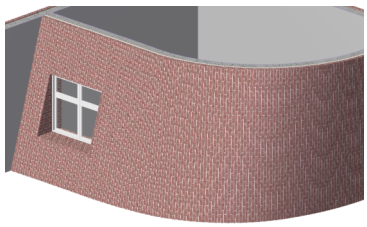
Click to show/hide the parameters.

Parameter	Description
Display Extents Below/ Above Cut Plane	Select to display the extents of the object below/above the cut plane
Range	Choose how much of the object to display
Infinite	Displays the entire object below/above the cut line
Finite Depth/Height	Displays the object below the cut line up to the specified depth/above the cut line up to the specified height

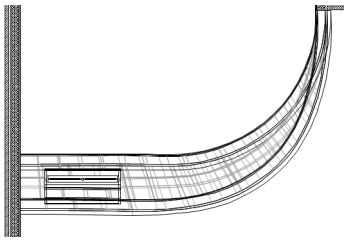
Parameter	Description
Class	Select the class to use from the current list of classes, select New to create a new class, or select <Contained Object Class(es)> to place the object below/above the cut plane in the same class(es) as the 3D contained object(s)
Fill	
None	Uses no fill for the area below/above the cut plane
Use 2D Attributes of Contained Object(s)	Uses the fill attributes of the original converted object(s) for the area below/above the cut plane
Use Class 2D Attributes	Uses the **Class** selection for the fill attributes of the area below/above the cut plane
Pen	
Use 2D Attributes of Contained Object(s)	Uses the pen attributes of the original converted object(s) for the area below/above the cut plane
Use Class 2D Attributes	Uses the **Class** selection for the pen attributes of the area below/above the cut plane
Dashed Hidden Line (Below Cut Plane only)	Select to display a dashed hidden line representation of the geometry below the cut plane  Mesh object in Top/Plan view  Top/Plan view of auto hybrid object with **Dashed Hidden Line** selected
Dash Style	When **Display Extents Below Cut Plane** and **Dashed Hidden Line** are selected, select the dash style for the hidden line display. The default dash style is the same as the document default for hidden line rendering.
Dash Shade	When **Display Extents Below Cut Plane** and **Dashed Hidden Line** are selected, select the dash shade for the hidden line display. The default dash shade is the same as the document default for hidden line rendering.
Include Hidden Lines (Above Cut Plane only)	Select to display the hidden lines of the geometry above the cut plane
Show as Reflected Ceiling Plan	When selected, displays all the auto hybrid geometry as if looking up, rather than looking down; applies to all tabs
3D Conversion Resolution	Sets the segmentation resolution used to display curved portions of the auto hybrid; this setting is independent of the **3D conversion resolution** Vectorworks preference and applies to all tabs
Smoothing Angle	When **Display Extents Below/Above Cut Plane** is selected, set a value to reduce the number of facets displayed in the geometry. For values greater than zero, facet lines are removed between any two faces of an object that are within that degree angle of each other; applies to all tabs.

Parameter	Description
Generate Intersecting Lines	When **Display Extents Below/Above Cut Plane** is selected, select to generate lines between intersecting geometry or deselect to omit intersecting lines; applies to all tabs. This option may make generating an auto hybrid object slower.
Save Current Settings as Defaults	Applies the current settings from all tabs as default parameters when creating auto hybrid objects

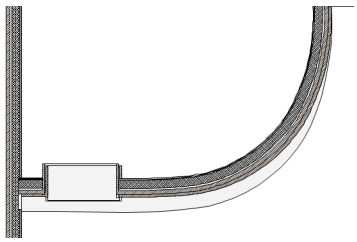
7. Click **OK**.



Modeled wall in 3D



Original plan view of modeled wall



Modeled wall converted to auto hybrid

Click here for a video tip about this topic (Internet access required).

Editing an Auto Hybrid Object

Editing an Auto Hybrid Object

The properties and 2D appearance of an auto hybrid object can be edited from the Object Info palette. You cannot edit the 3D geometry contained in the auto hybrid geometry directly; use the **Edit Auto Hybrid** command to edit the geometry.

To edit an auto hybrid object:

1. Select the auto hybrid object, and then select **Modify > Edit Auto Hybrid**.

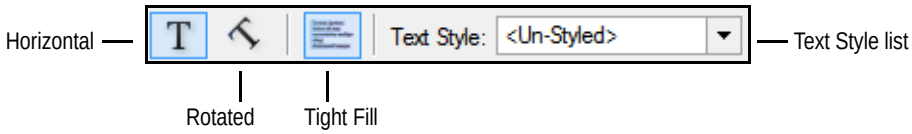
   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the auto hybrid and select **Edit** from the context menu, or simply double-click on the auto hybrid object.
2. The Edit window displays the item to be edited. A colored border around the drawing window indicates the editing mode is active.
3. Reshape the geometry, change the attributes, and perform any solid operations as needed.
4. Click the **Exit Auto Hybrid** button in the top right corner of the drawing window to return to regular drawing mode and update the auto hybrid.
5. On the Object Info palette, the elevation of the cut plane can be easily changed, and the auto hybrid can be set to display in 2D views only, or in both 2D and 3D views. Only auto hybrids with 3D geometry are exported to IFC.

Creating Auto Hybrid Objects
Object Editing Mode

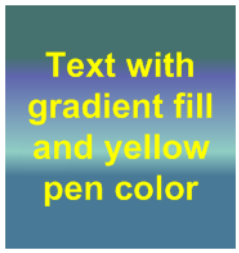
Text

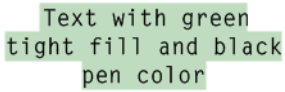
Inserting Text

Use the **Text** tool to create both single lines and blocks of text in your drawings. Once created, text objects can be moved, duplicated, duplicated in an array, and rotated. The text bounding box can be resized to adjust a block's height or width. Text is placed relative to the specified alignment point, not the bounding box.



Mode	Description
Horizontal	Creates horizontal text lines and text blocks
Rotated	Creates text blocks at an angle
Tight Fill	If a fill is applied to the text object, the fill displays only behind the text on any given line; blank lines have no fill
Text Style list	Applies a stored set of attributes to text as it is created, including the font, size, spacing, font style, alignment, and color





Setting the Default Text Attributes
Creating a Line of Text
Creating Text Blocks
Creating Rotated Text
Creating Text with a Tight Fill
Adding Tabs to Text
Pasting Text
Modifying Text
Checking Spelling
Finding and Replacing Text
Using Text Styles

Setting the Default Text Attributes

The default text attributes are applied to all text as it is added in the current drawing or any other. A text style applies a saved group of attribute settings. Select a text style as the default, or leave the text un-styled and set the default text attributes individually.

> Text styles can also be assigned to classes, to make it easier to create a consistent look for text that is embedded in dimensions, callouts, and other annotation objects.

To set the default text attributes:

1. Ensure that no objects or text are selected in the drawing.

If needed, click the selection arrow on an empty portion of the drawing.

2. Click the **Text** tool from the Basic palette.
3. From the Tool bar, set the **Tight Fill** mode as desired.
4. To set the default appearance of text, do one of the following:
   - From the Tool bar, select any **Text Style** other than <Un-Styled>.
   - From the Tool bar, select the <Un-Styled> **Text Style**. Then from the **Text** menu, either select **Format Text** to set several attributes from one dialog box, or select individual options as desired. See “Formatting Text” on page 389 for descriptions of the available options.

Creating a Line of Text
Creating Text Blocks
Creating Rotated Text
Creating Text with a Tight Fill
Using Text Styles

Creating a Line of Text

Use the **Text** tool to create a single line of text, such as a page header.

To type a single line of text:

1. Click the **Text** tool from the Basic palette, and select **Horizontal** from the Tool bar.
2. From the Tool bar, set the **Tight Fill** mode and **Text Style** as desired.
3. Click to designate the text insertion point; a text editing box with a blinking cursor displays.
4. Enter the text. Text wrapping is off by default, so the text box automatically expands as you type.
5. When the text line is complete, press the Esc key.

Modifying Text
Setting the Default Text Attributes
Creating Text Blocks
Creating Rotated Text
Creating Text with a Tight Fill
Using Text Styles

Creating Text Blocks

Use the **Text** tool to create a block of text, when more than a line of text is necessary.

To create a block of text:

1. Click the **Text** tool from the Basic palette, and select **Horizontal** from the Tool bar.
2. From the Tool bar, set the **Tight Fill** mode and **Text Style** as desired.
3. Click and drag to create a text box of the approximate width needed.

   A text editing box with a blinking cursor displays. The position of the text cursor indicates the horizontal alignment of the text to be placed.

4. Enter the text. Text wrapping is on by default, so the text automatically wraps to the next line when it reaches the edge of the text box. To end the current paragraph and start a new one in the current text block, press Enter.
5. When the text block is complete, press the Esc key.

Adding Tabs to Text
Modifying Text
Setting the Default Text Attributes
Creating a Line of Text
Creating Rotated Text
Creating Text with a Tight Fill
Using Text Styles

Creating Rotated Text

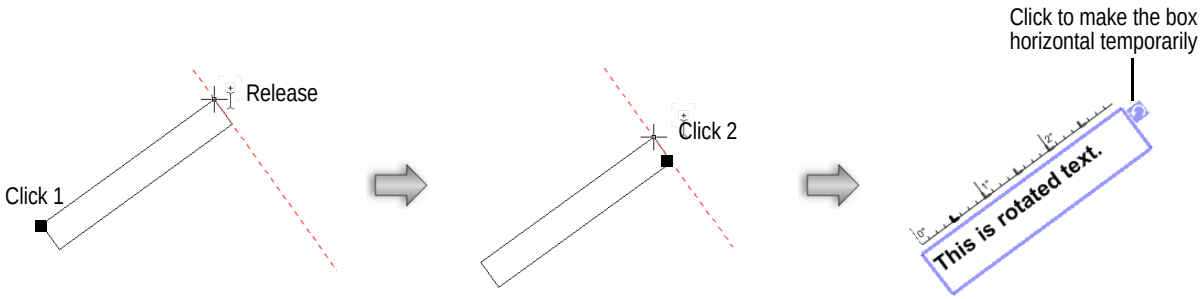
The Rotated mode of the **Text** tool creates text at an angle. Text objects can also be rotated after they are created; see “Rotating Objects” on page 1022 for details.

To create rotated text:

1. Click the **Text** tool from the Basic palette, and select **Rotated** from the Tool bar.
2. From the Tool bar, set the **Tight Fill** mode and **Text Style** as desired.
3. Click and drag the mouse to specify the angle and width of the text box, and release the mouse button to set. If needed, move the mouse to adjust the text box position along a line perpendicular to the text box. Click again to set the position.

   When drawing in rotated plan view (Vectorworks Design Series required), set angle snapping to snap to the plan rotation angle (see “Angle Snapping” on page 136) and display the Rotated Plan cue. If the text is created to the same angle as the plan, when the plan is unrotated, the text will align with the world coordinate system.
4. A text editing box with a blinking cursor displays.

   To always create and edit rotated text in a horizontal text box, select **Edit text horizontally by default** on the Edit tab of the Vectorworks preferences (see “Edit Preferences” on page 49).
5. Enter the text. The text automatically wraps to the next line when it reaches the edge of a text box. To end the current paragraph and start a new one in the current text block, press Enter.
6. When the text block is complete, press the Esc key.



Click and drag; release the mouse button to set the box width and angle

Move the mouse to adjust the box position along the red perpendicular line; click again to set the position

Enter the text

Adding Tabs to Text
Modifying Text

Setting the Default Text Attributes
Creating Text Blocks
Creating a Line of Text
Using Text Styles

Creating Text with a Tight Fill

The Tight Fill mode of the **Text** tool modifies the fill applied to a text object so that the fill displays only behind the text on any given line. Blank lines have no fill.

To create text with a tight fill:

1. Click the **Text** tool from the Basic palette, and select **Tight Fill** from the Tool bar.
2. From the Tool bar, select either the **Horizontal** or **Rotated** mode, and set the **Text Style** as desired.
3. From the Attributes palette, select the fill for the text object (unless the selected text style has a background fill).
4. Click and drag to create a text box of the approximate width needed.

   A text editing box with a blinking cursor displays. The position of the text cursor indicates the horizontal alignment of the text to be placed.
5. Enter the text. The text automatically wraps to the next line when it reaches the edge of a text box. To end the current paragraph and start a new one in the current text block, press Enter.
6. When the text block is complete, press the Esc key.

Text with a yellow tight fill on top of an oval with a pattern fill

Adding Tabs to Text
Modifying Text
Setting the Default Text Attributes
Creating Rotated Text
Using Text Styles

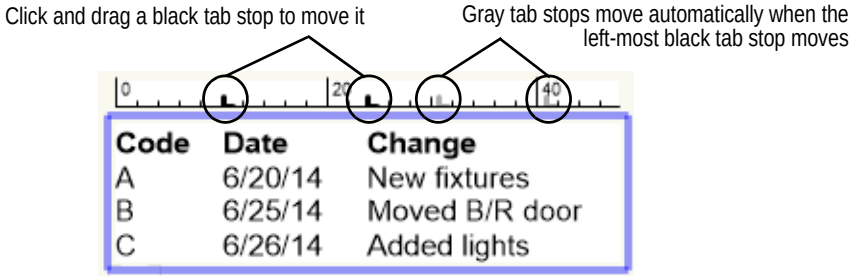
Adding Tabs to Text

Left-aligned tabs can be inserted into text during creation and editing.

To add tabs to text:

1. Click the **Text** tool from the Basic palette.
2. Enter or edit the desired text; press the Tab key to place tabs within the text where needed.
3. On the text box ruler, by default one tab stop is defined and displays with a black marker (at 0.5 inch for Imperial, or at 12.5 mm for Metric documents). Additional gray tab stops display at regular intervals to the right of the black tab stop. Adjust the tab stops as follows:
   - Click and drag a black tab stop to adjust its location.

- The spacing between the gray tab stops adjusts automatically as you move the left-most black tab stop. The gray tab stops display to the right of the right-most black tab stop.
- To have more control over the spacing between tab stops, click anywhere on the ruler to create one or more new black tab stops. Black tab stops can be moved independently.
- To remove a black tab stop, drag it off the ruler. If there is only one black tab stop, it cannot be deleted.



Creating Text Blocks
Creating Rotated Text

Pasting Text

To move text between Vectorworks files, select **Edit > Copy**, and then **Edit > Paste**. Text is added exactly as copied, including any formatting. If you define a text block before pasting, the text pasted into the block is wrapped to fit within the text block. If the scale of the original layer and the new layer are different, the text size changes accordingly; for example, if you copy 16 point text from a 1:1 layer and paste it to a 4:1 layer, the text changes to 4 points. This also means that any styled text you paste to a different layer scale becomes un-styled; reapply the text style to restore the proper text size.

When you paste text from a different program, click on the drawing using the **Text** tool first. If you paste text without first establishing an insertion point, each line of text is brought in as an individual text block. Embedded graphics are not supported and are removed when the text block is pasted into the Vectorworks file. In addition, multi-aligned text is converted to the current default alignment.

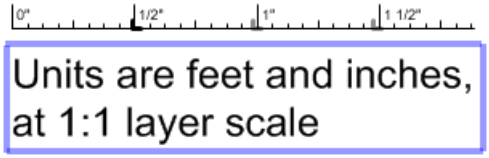
Modifying Text

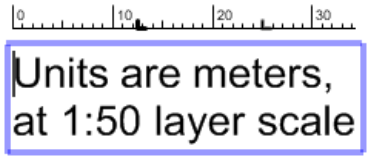
Once a text object is created, you may need to make changes, such as add words, change the font color, or change the margins. Changes can be applied to a portion of the text or to the whole object. A text object can even be converted into polylines and extruded, or converted to a 3D path object. Depending on what changes are required, either select the text object with the **Selection** tool or activate the text editing mode.

Text Editing Mode

To add, edit, or delete text in a text box, or to change the attributes of a portion of the text, the text editing mode must be active. To activate editing mode, double-click on the text object with the **Selection** tool, or click on it with the **Text** tool. Alternatively, right-click (Windows) or Ctrl-click (Mac) the text, and select **Edit** from the context menu.

Editing mode is indicated by a purple highlighted box with a ruler at the top. The units on the ruler reflect the current document units and layer scale.

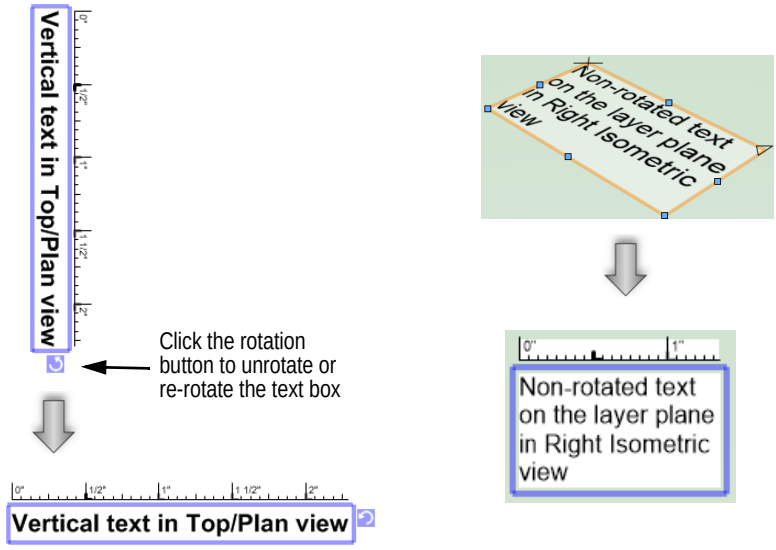




The standard keyboard shortcuts for Mac and Windows apply in the text editing box. For example, use Command+B (Mac) or Ctrl+B (Windows) to apply bold formatting.

- On Mac, and on Windows systems where GDI+ imaging is enabled, rotated text can be edited in place, or you can use the rotation button on the right side of the text editing box to display the text box horizontally for easier editing.
- If GDI+ imaging is disabled on Windows, rotated text automatically switches to a horizontal position in editing mode. The text box also switches position automatically if the current view is not Top/Plan.

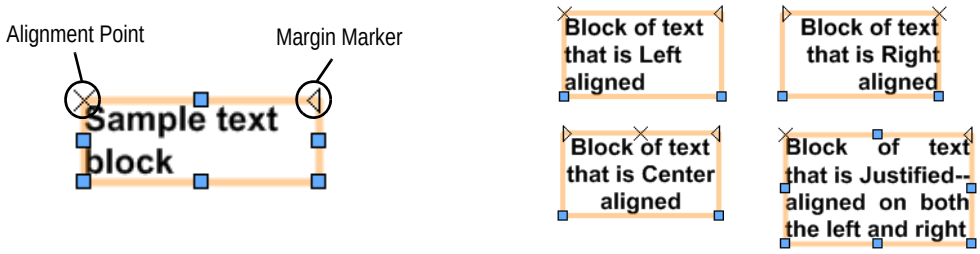
When the edits are complete, press the Esc key.



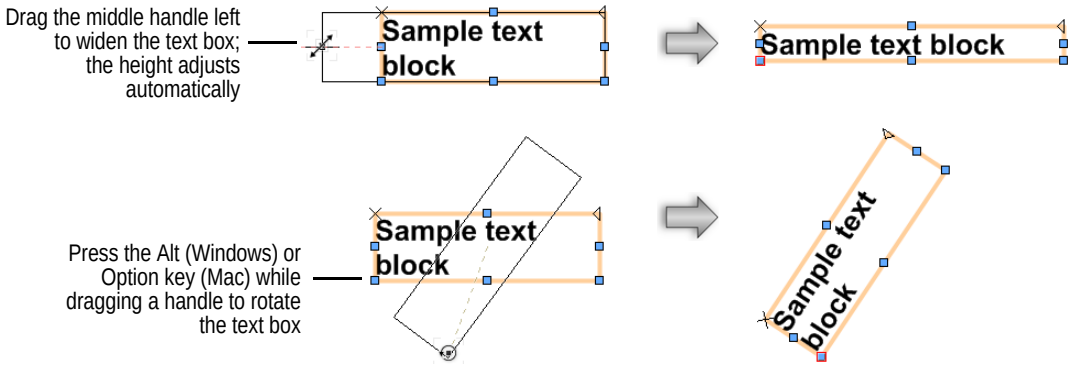
Editing Text Objects

To edit the properties of a text object (such as font or spacing), select it with the **Selection** tool. An X displays at the text alignment point. If the **Wrap Text** option is enabled, a triangular margin marker displays along the right, left, or both margins, depending upon the alignment setting.

Once the text object is selected, you can move it, or change the object's properties, such as alignment or text style.



The text object can also be resized and rotated similar to other 2D objects, using the reshape handles.



Formatting Text

The **Format Text** command modifies multiple text attributes from a single dialog box. Individual attributes also can be changed with various commands on the **Text** menu, including font, size, font style, alignment, spacing, and capitalization (lower case, UPPER CASE, and Title Caps). All attributes are available on the Object Info palette, as well. Some objects have the ability to select the font formatting while setting the object attributes.

Formatting can be applied to an entire text block or to selected characters and words. If no text is selected, the settings become the defaults for the **Text** tool.

To quickly apply the same set of text attributes to several objects, set up a text style. You can apply the text style to objects directly, or you can assign the text style to a specific class. See “Using Text Styles” on page 393 for details.

To format text:

1. To select the text to change, do one of the following:
   - To format the entire text box, select the text object with the **Selection** tool.
   - To format a word, double-click the text object to activate the editing mode; then double-click anywhere within the word to highlight it.
   - To format a line, double-click the text object to activate the editing mode; then triple-click anywhere within the line to highlight it.
   - To format a section of the text, double-click the text object to activate the editing mode; then highlight the desired text by dragging over it.

   To exit the text editing mode when text is highlighted, press the Esc key.
2. Once the text is selected, do one of the following:
   - From the **Text** menu, select the text option to change.
   - Select the option to change from the Object Info palette.
   - Select **Text > Format Text**. Alternatively, right-click (Windows) or Ctrl-click (Mac) the text, and select **Format Text** from the context menu. The Format Text dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Style	Displays the style of the currently selected item(s). If multiple items with different styles are selected, <Un-Styled> is selected. Select a style from the list; the remaining parameters change to match the settings for that style. If you change any of the other parameters after selecting a style, the style reverts to <Un-Styled>.
Save	Opens the Assign Name dialog box to save the current settings as a new style. Enter a name for the style and click **OK**. The new style is automatically assigned the current solid pen and fill color that are set as the default for new objects in the Attributes palette.
Font	Displays the font of the currently selected item(s) and lists all available, installed fonts. This field is blank when multiple items with different fonts are selected. Select a font from the list, or type the first letter(s) of the desired font to highlight the closest match in the list.
Size	Displays the size and unit measurement (points, millimeters, or inches) of the currently selected item(s). This field is blank when multiple items with different size/unit measurement are selected. Enter a size and select a unit measurement (points, millimeters, or inches).
Spacing	Displays the line spacing of the currently selected item(s). If multiple items with different spacing are selected, **Other** is selected. Select one of the standard spacing options, or select **Other** and specify a size and unit measurement (points, millimeters, or inches) of your own.
Font Style	Displays the font style of the currently selected item(s). If multiple items are selected and they have different settings for a particular style, the selection box displays an indeterminate state:  (Windows) or  (Mac). Specify the desired options. **Outline** and **Shadow** options are available for Mac only. **Superscript** and **Subscript** are disabled when the entire text block is selected.
Alignment	Displays the alignment of the currently selected item(s). If multiple items with different alignments are selected, the **Alignment** fields are blank. Specify the horizontal and vertical alignment.
Preview	Previews text font, size, and style

Recent Font List

Recently used fonts are stored for quick access. When you select **Text > Font**, the eight most recently used fonts display at the top of the font list (with the most recent font listed first), followed by an alphabetical list of all available fonts. In addition, when the Object Info palette contains a **Font** field, the top of the drop-down list is populated with the eight most recently used fonts, followed by an alphabetical list of the available fonts. Fonts changes made within the Format Text dialog box or Object Info palette also update the recent font list.

Transferring Formats with the Eyedropper Tool

Use the **Eyedropper** tool to quickly copy text formats from one text object and apply them to another object. See “Transferring Attributes” on page 1095.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Changing Other Text Features
Converting Text to Polylines
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Text Along a Path
Checking Spelling
Finding and Replacing Text
Using Text Styles

Changing Other Text Features

In addition to text format, other text features can be modified as needed.

- Use the **Capitalization** command to set all letters to lowercase, uppercase, or title case. Select the text and then select **Text > Capitalization**.
- Use the Attributes palette to apply color to text. Select the text and then choose a solid pen color.
- Use the Attributes palette to apply a fill to a text box. Select the text box and then choose a fill style and/or solid fill color. Add lines or spaces before and after the text to extend the box.
- Use the Attributes palette to apply opacity to both the text and fill of a text object. Select the text box and then choose an opacity setting.
- Use the Object Info palette to change the following text features.

Object Info Palette Field	Change
X and Y	Moves the text box along the X and Y axis
Width	Adjusts the text box width
Rotation	Rotates the text box
Wrap Text	Enables or disables text wrapping
Tight Fill	For filled text objects, enables or disables the tight fill option
Tracking	Adjusts the horizontal spacing between the selected characters. To change the spacing for all characters, select the text object. To change the spacing for specific characters, double-click the text object to enter editing mode, and then select the characters to change.

- Use one of various methods to rotate a text box. Select the text box and then use the **Rotate** command, **Rotate** tool, or **Selection** tool. See “Rotating Objects” on page 1022 for details.

Setting Special Text Sizes

To change text to a different standard size, you can select a size from either the **Text** menu or the Object Info palette. To set a non-standard text size, such as 60 pt, or 1 page inch, use the **Set Size** command.

To specify the text size:

1. Select **Text > Size > Set Size**. Alternatively, select Set Size from the **Size** list on the Object Info palette.

   The Set Text Size dialog box opens.
2. Select the desired unit of measure, enter the font **Size**, and then click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Attributes Palette
Formatting Text

Converting Text to Polylines

The **Convert Text to Polylines** command converts text created with TrueType or OpenType fonts into polylines. After it is converted, the text is no longer a font and can be edited just like any other polyline. This is useful for creating 3D
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

text objects. By their nature, TrueType and OpenType fonts are defined by Bézier curves and arc points. These same definitions are used to convert the text objects.

The text conversion is not affected by the conversion resolution setting in the Vectorworks Preferences dialog box. However, if the converted polyline is extruded, the 3D resolution setting in this same dialog box affects how the polyline is extruded.

To convert text to polylines:

1. Select the text (lines or blocks) to convert.
2. Select **Text > Convert Text to Polylines**.

   The text is converted into a group of polylines.
3. To extrude the polylines, creating 3D text, select the group of polylines. Ungroup them by selecting **Modify > Ungroup**.
4. Select all of the individual polylines and select **Model > Extrude**.

   The Create Extrude dialog box opens.
5. Specify the extrusion length and click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Text Along a Path

Creating Text Along a Path

The **Text Along Path** command converts text created with TrueType or OpenType fonts into polylines, and places them along a specified path. The appearance of the path object can be adjusted after placement.

To create 3D text along a path:

1. Select the text and a path object. The text must be a single line, and the path must be long enough for the text, or text along path conversion will not occur.
2. Select **Text > Text Along Path**.

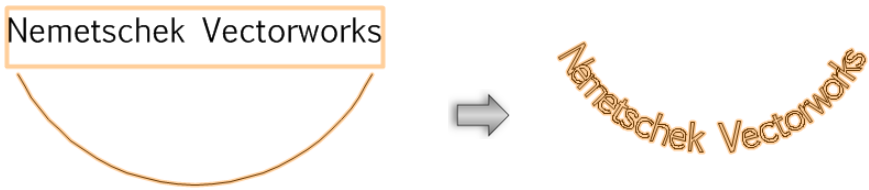
   The Text Along Path dialog box opens. Specify the options for creating the text along the path.

Click to show/hide the parameters.

Parameter	Description
Text Size	Specifies text size options
Preserve Height and Width	Maintains the aspect ratio of the text, keeping width and height parameters the same as they were in the original text
Scale Width Only	Changes the width of the text to fit the path, but does not change the height accordingly (resulting in wider, shorter text, depending on the path)
Scale Height and Width	Changes the width of the text to fit the path, and then changes the text height to match (resulting in wider, tall text, depending on the path)
Create Text As	Specifies the format for converting the text
Curves	Converts the text into a group of polylines (if the path is drawn on the active layer plane) or NURBS curves (if the path has a Z height or a **Rot about Path** greater than 0)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Surfaces	Converts the text into a group of NURBS surfaces
Extrusions	Converts the text into a group of extrude objects; specify the **Height** of the extruded letters

3. Click **OK**. The selected text follows the path object, and the original path object is deleted.



The Text Along Path parameters can be edited in the Object Info palette. The parameters are identical to those in the Text Along Path dialog box, with two additional parameters.

Click to show/hide the parameters.

Parameter	Description
Above Path	Places the bounding box of the letters directly above the path; "above" depends on the direction that the path was drawn. Deselect this option to place the letters "below" the path. Depending on the path and letters, the appearance of the text may be improved by switching it above or below the path.
Rot About Path	Indicates the angle of rotation about the path, using the path as a rotation axis

To edit the path object, select **Modify > Edit Text Along Path**, and then select Path. The path object can be edited with the **Reshape** tool. To reverse the direction of the path object, click **Reverse Direction**; this affects the text placement above or below the path.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Converting Text to Polylines

Using Text Styles

A text style is a resource that specifies text attributes, including the font, size, line spacing, font style, text alignment, and text and background colors. Text styles make it easy to apply a consistent look to all text throughout a file; if you change a text style, all objects that use the style are updated at once. A text style can be applied to objects created with the **Text** tool, as well as to the text portions of other objects, such as dimensions, callouts, and other annotation objects.

Not all features or plug-in objects can make use of a text style. For example, worksheets and space objects have text components, but they cannot use text styles.

A text style can also be part of a class definition, so that all items in that class have the same style. This makes it easy to apply the same text style to all dimensions, for example. See "Setting Class Properties" on page 179.

Some basic text styles are provided with the Vectorworks program as default content; these display on the **Text Style** list on the Tool bar when the **Text** tool is active. In addition, you can create custom text styles for your own use, or to share with coworkers. Because they are resources, text styles can be exported and imported into another file, copied among files, and shared via workgroup referencing. For more information about default content, see "Resource Libraries" on page 219. For more information about using custom resources, see "Accessing Existing Resources" on page 229 and "Creating Resource Libraries" on page 220.

Click here for a video tip about this topic (Internet access required).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Text Styles

Create custom text styles as needed. These are saved with the current file, and they can also be exported into other files or added to resource libraries.

To create a new text style:

1. Do one of the following:
   - In the Resource Browser, select **Resources > New Resource**; select **Text Style** from the list of resource types.
   - Click the **Text** tool from the Basic palette, and select New from the **Text Style** list on the Tool bar.

   The Create Text Style dialog box opens.

   Alternatively, select **Text** > **Format Text** to open the Format Text dialog box. Set the parameters as needed, click **Save**, and enter a name for the new style. See “Formatting Text” on page 389.

Click to show/hide the parameters.

Parameter	Description
Style Name	Enter a name for this text style
Font	Select a font from the list, or type the first letter(s) of the desired font to highlight the closest match in the list
Size	Enter a size and select a unit measurement (points, millimeters, or inches)
Spacing	Select one of the standard spacing options, or select **Other** and specify a size and unit measurement (points, millimeters, or inches) of your own
Font Style	Specify the desired options. **Outline** and **Shadow** options are available for Mac only
Alignment	Specify the horizontal and vertical alignment
Color	Select a color for the **Text** (see “Selecting a Color from a Color Palette” on page 1135). To have a background fill behind the text, select the **Background** option and also specify a color. To have no background fill, deselect the **Background** option.
Preview	Previews text font, size, style, text color, and background color, if applicable

2. Click **OK** to create the resource.

   The new text style is saved with the file and displays in the Resource Browser under Text Styles. Text styles also display on the Tool bar when the **Text** tool is active, and on the Object Info palette when an object that uses text styles is selected.

Applying Text Styles

Text styles can be applied from several different places in the Vectorworks program. They can be applied to text objects, or to the text portions of several other objects, such as dimensions, callouts, and other annotation objects.

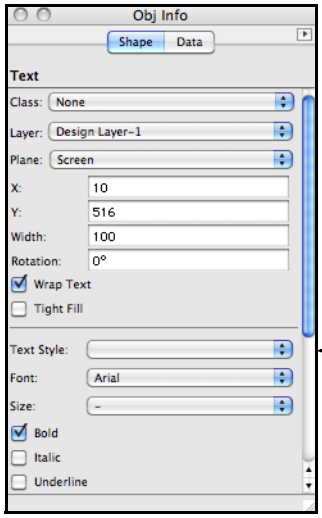
- From the Resource Browser, drag and drop a text style onto an object.
- Select all or part of a text object, or another object that uses text styles, and select a **Text Style** from the Object Info palette.
- Add a text style to a class definition, and then apply that class to an object.
- Select all or part of a text object, or another object that uses text styles, select **Text > Format Text**, and then select a **Style** from the Format Text dialog box.

- With no objects selected, click the **Text** tool, and select a **Text Style** from the Tool bar. This sets the default text style that will be used when new objects are created.

Other Features of Text Styles

- Although text style resources can include a color, some types of objects do not use the text style color. For example:
  - Text, callouts, and general notes always use the text style color.
  - Dimensions and other annotation objects always use the current pen color.
- If an object's text style comes from its class definition, the text style displays as <Class Text Style>.
- Text with no style applied displays <Un-Styled> as the text style. If you change any of the attributes of styled text (for example, if you change the size from 16 points to 18 points), the text style automatically becomes <Un-Styled>. Text also becomes un-styled if you apply the <Un-Styled> option to it, but the text attributes remain the same.
- If you move styled text to a layer that has a different scale from the original layer, the text will be scaled accordingly, and it automatically becomes <Un-Styled>; reapply the text style if you want to restore the original text size.
- The alignment, spacing, and background color attributes of a text style are only applied if an entire text object is selected; these attributes are ignored if you apply a style to a portion of a text object.
- If a text object contains text with multiple styles, the text object itself does not display a text style in the Object Info palette. The styled portions of text within the object are still updated if the text style resource is changed.

Green text is style 1 and blue text is style 2



Because it contains two text styles, the text object itself displays no Text Style

Editing Text Styles

If you change a text style resource, all objects in the file that use the style are updated at once.

To edit a text style:

1. From the Resource Browser, select the text style, and then select **Resources > Edit**.

   The Edit Text Style dialog box opens.

2. Edit the parameters as described in "Creating Text Styles" on page 394 and click **OK**.

   The text style definition and all objects in the file that use the text style are updated.

Inserting Text

Modifying Text

Checking Spelling

Check the spelling of either selected text or all the text in a file with the **Check Spelling** command. Spelling in text blocks, symbols, records, worksheets, and viewports can be checked. Available dictionaries include:

- Danish
- Dutch
- English (American)
- English (British)
- Finnish
- French
- German
- Italian
- Norwegian
- Portuguese (Brazilian)
- Portuguese (Iberian)
- Spanish
- Swedish

Dictionaries can be edited and added.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Checking the Spelling
Correcting Spelling Errors
Spelling Check Options
Adding and Editing Dictionaries-

Checking the Spelling

To check the spelling of a selected object:

1. Select the object.
2. Select **Text > Check Spelling**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) the text, and select **Check Spelling** from the context menu.
3. If a spelling error is detected, the Selection Spelling Check dialog box opens so that corrections can be made.

To check the spelling of all text in the file:

1. Ensure that no objects are selected.
2. Select **Text > Check Spelling**.

   The Spelling Check Filter dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Text Blocks	Checks text contained in text blocks
Symbols	Checks text contained in symbol definitions
Records	Checks text contained in records
Worksheets	Checks text contained in worksheets
Viewports	Checks text annotations contained in viewports

3. Select the items to have spelling checked, and then click **OK**. If a spelling error is detected, the Document Spelling Check dialog box opens so that corrections can be made.

   If no spelling errors are detected, a message displays to indicate that the spelling check is complete.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Correcting Spelling Errors

If a spelling error is found, either the Selection Spelling Check (when checking a selection) or the Document Spelling Check (when checking all text) dialog box opens. Both dialog boxes contain the same options.

Click to show/hide the parameters.

Parameter	Description
Spelling error in	Identifies the location of the object containing the potential spelling error
Not in Dictionary	Lists the potentially misspelled word; if none of the suggested corrections in the **Suggestions** list is an acceptable replacement, type the correction into the **Not in Dictionary** field (or delete the word by leaving the field blank). Then click **Change** or **Change All** to replace the error with the typed word. The word is replaced and the spelling check resumes.
Suggestions	Suggests the closest matching word(s) from the dictionary
Change / Change All	Select one of the suggested words from the list of **Suggestions** to replace the misspelled word and click **Change**. The misspelled word is replaced with the suggested word. Alternatively, press Enter with the suggestion selected. To replace all occurrences of the same error in the file, click **Change All**. The word is replaced and the spelling check resumes.
Ignore / Ignore All	If the word is spelled correctly, but it is not present in the dictionary, click **Ignore** to leave the word as is and continue the spelling check. Click **Ignore All** to ignore all occurrences of the word in the file. The word is ignored and the spelling check resumes.
Learn	Click to add the word to the dictionary; this allows the spelling checker to recognize all future occurrences of the word
Options	Click to customize the spelling check options; see “Spelling Check Options” on page 397
Done	Click to discontinue the spelling check; all changes up to that point are saved, but can be undone by selecting **Edit > Undo**

The spelling of layers, classes, symbol names, object names, script palette names, dimension text, locked objects, or records attached to locked objects is not checked.

Spelling Check Options

Click the **Options** button in the Spelling Check dialog box to customize the spelling check function. The Spelling Check Options dialog box opens.

Specify the types of misspelled words for the spelling checker to find. If a check box is not selected, the spelling checker ignores errors for that category of words. Examples include:

- Capitalized words: Canada
- Words with mixed case: VectorScript
- Words in all caps: ANGLE
- Words with numbers: Q4

Customize the spelling checker to reduce unnecessary spelling checks in your typical files. Click **OK** to return to the Selection Spelling Check or Document Spelling Check dialog box.

Adding and Editing Dictionaries

The user dictionary, UserDictionary.txt, is a text file located in [Vectorworks]\Plug-Ins\Dictionaries. It can be edited, if desired. When manually editing a dictionary, type the word followed by a tab and the letter "i" to indicate that the spelling checker should ignore the word.

Additional dictionaries, such as a foreign language dictionary, can be added by placing the dictionary file into the [Vectorworks]\Plug-Ins\Dictionaries folder. The spelling checker automatically uses all the dictionaries with the .clx extension in the folder to check the spelling. However, the addition of multiple dictionaries can slow down the spelling check process.

Click here for a video tip about this topic (Internet access required).

Finding and Replacing Text

The **Find-Replace Text** command searches for and optionally replaces text strings within a Vectorworks file. It can search for and replace text strings within text objects, record fields, and worksheet cells.

Use this command to find a text item, replace it, search for more occurrences, or replace all occurrences. All settings are retained from one use to the next, including find and replace text strings.

To find-replace text:

1. Select **Text > Find-Replace Text**.

   The Find or Replace Text dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Action	Locates and, if selected, replaces a given text string with a new text string
Find Next	Finds the next occurrence of the text string
Replace Next	Replaces the next occurrence of the text string
Replace All Selected	Replaces all selected occurrences of the text string
Replace All	Replaces all occurrences of the text string
Look in	Searches for occurrences in the specified parts of the document
Text Objects	Searches in all text objects
Record Fields	Searches in all record fields, including Callout objects
Worksheets	Searches in all worksheets; appears dimmed if **Replace All Selected** is chosen, since there is no selection attribute for a worksheet
Find String	Enter text string to search for
Replace With	Enter replacement text string; dimmed if **Find Next** is selected
Options	Specifies the depth of the search
Active Layer Only	Searches in the active layer only
All Layers	Searches on all layers within the document, regardless of visibility
Visible Layers Only	Searches in all currently visible layers
Case-sensitive	Searches for text that exactly matches the criteria, including capitalization

2. Enter the desired search and, if using, replace criteria.
3. Click **Find/Replace**.

Creating Hyperlinks

The **Hyperlink** tool inserts a hyperlink object in the drawing, for navigating to saved views or sheet layers in the file, opening other files or folders, or launching applications or web sites. The hyperlink can consist of text and/or geometry, and can be created from a custom symbol; symbols can be page based or world based. Hyperlink properties are easy to customize and they can be batch edited from a worksheet. Hyperlinks to websites and sheet layers can be exported to a PDF file (Vectorworks Design Series required); hyperlinks to sheet layers are functional when the sheet layers are exported as a batch to a single PDF file.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Hyperlink

To insert a hyperlink object:

1. Click the **Hyperlink** tool from the Dims/Notes tool set.
2. Click **Preferences** from the Tool bar to specify the hyperlink parameters.

   Alternatively, double-click on the **Hyperlink** tool.

   The Hyperlink Settings dialog box opens. The parameters in the **Details** area depend on the selected **Function**.

Click to show/hide the parameters.

Parameter	Description
Layout	Select the desired appearance of the hyperlink
Symbol	Select a symbol from the hyperlink symbols available in the file and the default content symbols provided. (Default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219.) Symbols can be page based or world based; default hyperlink symbols are page based. #Hyperlinktext# indicates the area of the symbol that displays the **Label** text.
Edit Symbol	The selected symbol can be edited for use in this file. Clicking **Edit Symbol** exits the Hyperlink Settings dialog box, and the Edit window displays for editing the symbol’s 2D component (a colored border around the drawing window indicates the editing mode is active). Reshape the geometry, change attributes, add text, paste images, and perform other editing operations, and then click **Exit Symbol** to return to the drawing. The hyperlink symbols in the drawing can also be edited from the Resource Browser; see “Editing Symbol Definitions” on page 247. Edited symbols are available for selection from the **Symbol** list.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
New Symbol	Opens the New Symbol dialog box; provide a name for the new hyperlink symbol and click **OK** to return to the Hyperlink Settings dialog box. A new text-only hyperlink symbol is automatically created and added to the resources in the file. This new custom hyperlink symbol can then be selected from the **Symbol** list and edited by clicking **Edit Symbol**.
Details	
Label	Enter the text to display for the hyperlink
Function	Select the hyperlink function
Open Webpage	Launches the specified website in the default browser; enter the URL in **Webpage**
Launch Application	Launches the specified application; click **Browse** to select the application. Choose whether the path to the application should be saved as absolute or relative. To set a relative path, the Vectorworks file must exist on the same volume as the application.
Open Document	Opens a file, such as a movie or image; click **Browse** to select the file. Choose whether the path to the file should be saved as absolute or relative. To set a relative path, the Vectorworks file must exist on the same volume as the file to open.
Open Folder	Opens the specified folder; click **Browse** to select the folder. Choose whether the path to the folder should be saved as absolute or relative. To set a relative path, the Vectorworks file must exist on the same volume as the folder to open.
Activate Saved View	Displays the specified saved view; select the view from the **Saved View** list
Activate Sheet Layer	Activates the specified sheet layer; select the sheet layer from the **Sheet Layer** list

3. Select the hyperlink symbol and function, and then click **OK**.

   On the Tool bar, the current hyperlink function and target displays.

4. Click in the drawing to insert the hyperlink object.
5. To execute the hyperlink, Ctrl-click (Windows) or Cmd-click (Mac) on the hyperlink, or click **Open Link** from the Object Info palette or context menu of a selected hyperlink object.

~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Hyperlinks
Editing Symbol Definitions

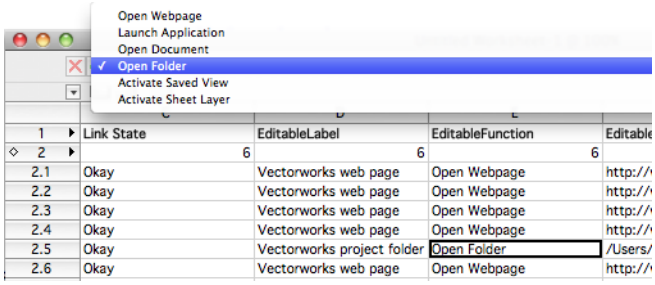
Editing Hyperlinks

The Object Info palette of a selected hyperlink object displays its currently assigned **Function** and **Target**. Click **Open Link** to execute the hyperlink's function. Click **Edit** to open the Hyperlink Settings dialog box and specify the hyperlink properties as described in "Inserting a Hyperlink" on page 399. Other methods of accessing the Hyperlink Settings dialog box include double-clicking on the hyperlink or pressing Ctrl-click (Mac) or right-click (Windows) and selecting **Edit** from the context menu.

If a link is invalid or broken, the Object Info palette displays a message in red text, and an alert displays when the hyperlink is executed. Edit the hyperlink settings to specify a valid target.

The hyperlink symbol definition can be edited from the Resource Browser (see "Editing Symbol Definitions" on page 247); this affects all instances of the symbol.

It can be tedious to edit multiple hyperlinks in large projects. In this type of situation, use scripted commands or create a report containing the hyperlink objects by selecting **Tools > Reports> Create Report**. See "Creating Reports" on page 1316. In the resulting worksheet, columns labeled "Editable" can be changed, for convenient editing.
~~~~~~~~~~~~~~~~~~~~~~~~~~



	Link State	EditableLabel	EditableFunction	Editable
2	6	6	6	
2.1	Okay	Vectorworks web page	Open Webpage	http://(
2.2	Okay	Vectorworks web page	Open Webpage	http://(
2.3	Okay	Vectorworks web page	Open Webpage	http://(
2.4	Okay	Vectorworks web page	Open Webpage	http://(
2.5	Okay	Vectorworks project folder	Open Folder	/Users/
2.6	Okay	Vectorworks web page	Open Webpage	http://(

D Exporting Hyperlinks to PDF

Only hyperlinks that open a web page or navigate to a sheet layer are functional when exported to a PDF file.

To obtain successful hyperlinks to sheet layer targets, the file must be exported with the **File > Publish** command. The sheet layer containing the hyperlink, as well as the target sheet layers, need to be exported to a single PDF file so that the links can be executed. **Export PDFs as separate files** must be deselected in the Publish dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Hyperlink
Editing Symbol Definitions
Exporting PDF Files
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Space Planning

The Vectorworks Architect product performs space planning and programming studies, and creates schematic floor plans.

These features can be combined in various ways depending on your preferred workflow:

- To begin the design process, draw the spaces. Reposition and reshape the spaces as needed to develop a schematic floor plan. Then create the walls automatically from those spaces.
- Begin with a solid model, and then create the exterior walls from the model.
- Create the walls first, and then create the space objects automatically to determine the areas enclosed by the walls.
- Using space programming, import an adjacency matrix that was provided by a client, and automatically create a bubble diagram and a stacking diagram. Reshape and reposition the space objects in the bubble diagram to create a floor plan, and then create the walls automatically from the spaces.
- Create the initial schematic design with polylines instead of spaces, and then convert the polylines to spaces.
- Create the model with stories, and associate the net/gross volume of the spaces along with the stories.

Use the **Space** tool to create a schematic floor plan. The Vectorworks Architect product can then automatically create walls from those spaces. To track room finish data on a schedule, add finish information to the spaces. If necessary, IFC data (including extended space properties used by the General Services Administration) can be attached to the spaces. The net and gross volume of a space can be associated with story elevation, so that as story layer levels change, the space volume adjusts accordingly.

If walls or polylines that represent spaces already exist in the drawing, use space planning commands to create spaces from the walls or polylines.

When creating and updating spaces, the visibility of walls can affect the spaces, depending on the situation and action:

- When creating spaces, only visible walls are taken into account
- When updating existing spaces (clicking **Update Boundary** from the Object Info palette), only visible walls are taken into account
- When existing spaces are regenerated, for example due to changed space labels, a new height value, or moved walls, wall visibility has no effect on automatically bounded spaces, because the spaces are already associated with the walls.

See “Editing Space Boundaries” on page 434 for more information.

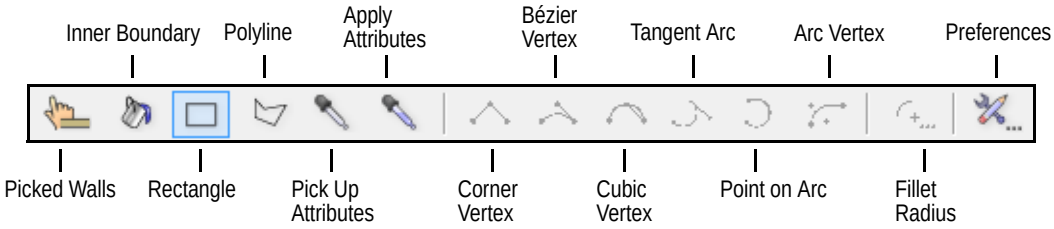
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Spaces with the Space Tool

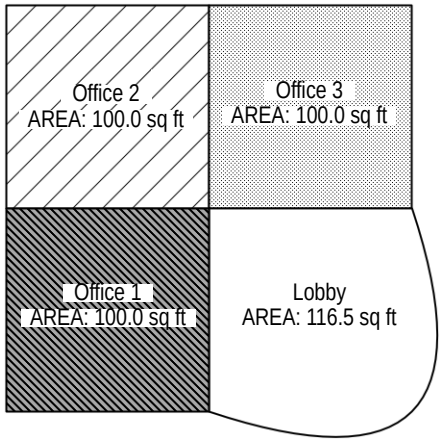
Spaces are path objects that have the characteristics of a building space or room, which include information such as the space name, number, finish information, and height. The space label is drawn using a pre-defined or user-defined symbol, can display not only name and number, but virtually any desired information. Space objects automatically calculate their area, volume, and perimeter.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The **Room Name Simple** tool (in the Dims/Notes tool set) creates a text label only; it does not attach room finish, area, IFC, or GSA information to a space object.

To draw a free-standing space with the **Space** tool, select either the Rectangle or the Polyline mode. If the walls are already drawn, select the Picked Walls mode or Inner Boundary mode to create a space bounded by visible walls.



Mode	Description
Picked Walls	Creates a space object based on a closed set of selected walls
Inner Boundary	Creates a space object in a clicked area that is bounded by visible walls
Rectangle	Draws a rectangular space object
Polyline	Draws a polyline space object; as with a polyline, select one of six types of control points for the vertices from the Tool bar
Pick Up Attributes	Picks up the attributes of a space that is clicked upon. The properties that are transferred can be customized; see “Space Settings: Advanced Settings Pane” on page 416 for details.
Apply Attributes	Transfers the space attributes that have been picked up with the Pick Up Attributes mode to a space that is clicked upon. Press the Ctrl key (Windows) or the Option key (Mac) when you click to toggle between the Pick Up Attributes and Apply Attributes modes.
Corner Vertex	For Polyline mode, creates the space using polyline segments with straight lines and angled vertices at the control points
Bézier Vertex	For Polyline mode, creates the space using polyline segments with curves pulled toward, but not touching the control points
Cubic Vertex	For Polyline mode, creates the space using polyline segments with curves that pass through the control points
Tangent Arc	For Polyline mode, creates the space using polyline segments that are tangent to the previous segment
Point on Arc	For Polyline mode, creates the space using polyline segments that are drawn by clicking three points: the start point, a point the arc passes through, and the end point
Arc Vertex	For Polyline mode, creates the space using polyline segments with curves that look like a fillet placed at the control points
Fillet Radius	For Polyline mode, sets the fillet radius when the Arc Vertex mode is selected
Preferences	Sets the default parameters that are used for each new space object



Creating Spaces from Existing Walls

Spaces can be automatically created by first selecting the enclosing walls or by clicking within the boundary of a closed set of visible walls. The spaces are automatically associated with their bounding walls.

Alternatively, select **AEC > Space Planning > Create Spaces from Walls** to create spaces from a set of enclosed walls on a specified design layer (see "Creating Spaces from Walls" on page 430).

To create a space within a selected set of visible walls:

1. Click the **Space** tool from the Space Planning tool set or the Building Shell tool set.
2. From the Attributes palette, set the 2D attributes for the space (fill, pen, opacity, and line thickness).
3. From the Tool bar, select Preferences. Set the default parameters for the space objects.
4. From the Tool bar, select Picked Walls mode.
5. Click on each wall that forms the wall set and press Enter, or click the check mark button on the Tool bar. The space is created with the attributes and settings you specified.
6. Add information specific to the space.

To create a space within a closed set of visible walls:

1. Click the **Space** tool from the Space Planning tool set or the Building Shell tool set.
2. From the Attributes palette, set the 2D attributes for the space (fill, pen, opacity, and line thickness).
3. From the Tool bar, select Preferences. Set the default parameters for the space objects.
4. From the Tool bar, select the Inner Boundary mode.
5. Click in an open area of the drawing that is bounded by walls; the walls must be visible, joined together, and on layers that have the same layer scale as the space layer. The space is created with the attributes and settings you specified.
6. Add information specific to the space.

Drawing Spaces with the Space Tool

To draw a space with the **Space** tool:

1. Click the **Space** tool from the Space Planning tool set or the Building Shell tool set.

2. From the Attributes palette, set the 2D attributes for the space (fill, pen, opacity, and line thickness).
3. From the Tool bar, select Preferences. Set the default parameters for the space objects.
4. From the Tool bar, either select the Rectangle mode, or select the Polyline mode and then select the type of control point for the polyline vertices.
5. Draw the space as follows:
   - For a rectangular space, click to begin the rectangle, and then click again to finish the rectangle and create the space.
   - For a polyline space, click to begin the polyline, and then click to set each polyline vertex. Click on the start point to end the polyline and create the space.

   The space is created with the attributes and settings you specified.
6. Add information specific to the space.

Adding Information to Spaces

After all spaces have been created, select specific spaces and add more information to them on the Object Info palette, such as a space name and room finishes. The items that are changed most frequently are available on the Shape tab. To access the full set of space properties, click the **Settings** button to open the Space Settings dialog box.

> The properties that are available on the Object Info palette are determined by the Advanced Settings pane on the Space Settings dialog box. See “Space Settings: Advanced Settings Pane” on page 416 for details.

Copying Attributes to Other Spaces

If several spaces need to have attributes in common, such as the same space name and room finishes, copy the attributes from an existing space and apply them to the other spaces.

To copy attributes to other spaces:

1. Click the **Space** tool from the Space Planning tool set or the Building Shell tool set, and then from the Tool bar, select Preferences.
2. From the Space Settings dialog box, select the Advanced Settings pane.
3. Select the attributes to transfer between spaces from the Eyedropper Transfer Properties window (see “Space Settings: Advanced Settings Pane” on page 416 for details).
4. Select the Pick Up Attributes mode from the Tool bar. Click the space with the attributes you want to transfer.
5. Select the Apply Attributes mode from the Tool bar, and click each space that should have those attributes.

   Alternatively, press the Ctrl key (Windows) or the Option key (Mac) while you click to activate the Apply Attributes mode.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Space Settings
Editing Space Boundaries
Adding Information to Spaces

Space Settings

Use the **Space** tool’s Preferences to set the default parameters for space objects before you create them, including the label style, automatic numbering, boundary calculations, and 3D graphic attributes. For maximum control over the
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

display of spaces, you can set default classes for the label, label leader line, 2D attributes, 3D attributes, and the entire space object. Preference settings are an easy way to apply common, consistent settings to all spaces.

The space's 2D graphic attributes are not taken from the Preferences settings. Instead, set the appropriate fill and pen styles in the Attributes palette before you create the spaces.

Once the spaces have been created, use the Object Info palette to edit the properties of individual spaces, including space name and occupant, room finishes, and any additional data attached to the space. The items that are changed most frequently are available on the Shape tab. To access the full set of space properties, click the **Settings** button to open the Space Settings dialog box.

The properties that are available on the Object Info palette are determined by the Advanced Settings pane on the Space Settings dialog box. See "Space Settings: Advanced Settings Pane" on page 416 for details.

To reduce the time required to regenerate space objects after properties are edited, Vectorworks only regenerates the space components that are changing and only recalculates the bounds of the spaces if necessary. To force the complete regeneration of edited space objects, select **Tools > Utilities > Reset all Plug-Ins.**

The fields that display on the Space Settings dialog box are slightly different, depending on whether you access the dialog box with the Preferences mode of the **Space** tool or with the **Settings** button on the Object Info palette.

The settings are grouped into several panes of related parameters. Select each group of parameters from the list in the left pane of the dialog box; the parameters display in the center pane.

The right pane of the dialog box always displays a preview of the space label(s) based on the current settings. To edit the appearance of any of the space labels (for example, to change the font size or the pen color), click the label's **Edit Layout** button at any time.

A few controls can be edited directly from the Object Info palette, as described in "Space Properties" on page 417.

Space Settings: Preview

Click to show/hide the parameters.

Parameter	Description
Space Label Preview	Dynamically displays a preview of the space label(s) with the currently selected parameters; see "Space Settings: Space Label 1, 2, and 3 Panes" on page 412 Space Label 1 previews by default. Space Label 2 and Space Label 3 preview only if **Enable multiple labels** is selected on the Advanced settings pane and **Display Space Label** is selected on the Space Label 2/3 pane.
Edit Layout	Click to enter object editing mode to edit the format of the specified label; see "Object Editing Mode" on page 1004

Space Settings: Numbering Pane

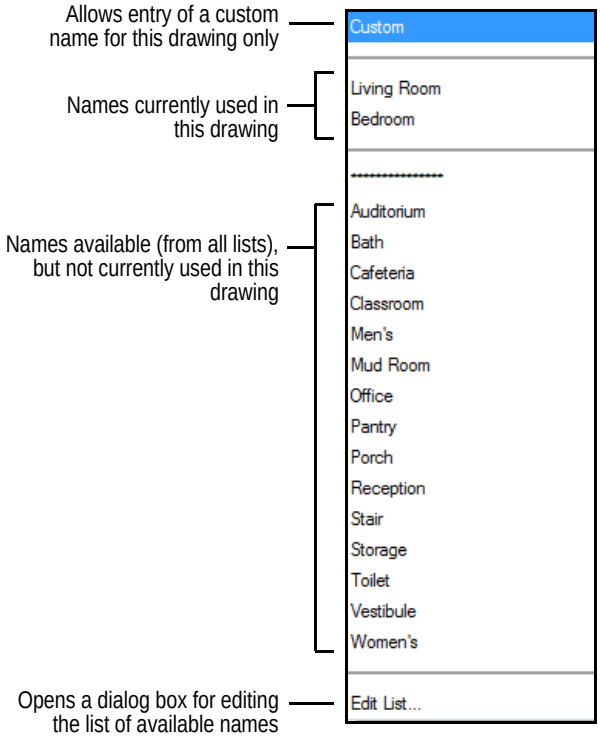
Click to show/hide the parameters.

Parameter	Description
Space Number	When you set the default preferences for the **Space** tool, if the **Space Number Style** is Manual, leave this field blank and add the numbers to spaces after they are created. For all other numbering styles (auto-numbering), the field displays the **Next value** and cannot be edited. When you edit a space using the **Settings** button from the Object Info palette, if the **Space Number Style** is Manual, enter a number for this space. For all other numbering styles (auto-numbering), the field displays the space's assigned number and cannot be edited.
Space Number Style	Specifies the type of numbering to use on space objects. • Select Manual to enter a number for each space manually in the **Space Number** field. • Select Counter Only to have the Vectorworks program automatically assign the next available number to each space as it is created. • If custom number styles were created, they display at the bottom of the list; select the style to use to auto-number each space as it is created.
Edit Number Styles	Opens the Edit Number Style dialog box to create or edit number styles. See "Custom Number Styles" on page 418 for details.
Auto-Number Settings	Specify the following values if auto-numbering is in use (that is, **Space Number Style** is not set to Manual)
Start value	Specifies the numeric value to assign to the first new space
Increment	Specifies the amount to increment the numeric value of each new space; this number is automatically added to the **Next value** each time a space is created
Next value	Specifies the numeric value to assign to the next new space that is created
Validate Auto-Numbering	Available only when you edit a space using the Settings button from the Object Info palette (not when you set the default preferences for the **Space** tool); opens the Validate Auto-Numbering dialog box to adjust the automatic space numbers. See "Validating Auto-Numbering" on page 419 for details.

Space Settings: Occupancy Pane

Click to show/hide the parameters.

Parameter	Description
Space Type	Identifies a space as either Normal (for example, a room) or Full Floor (the entire floor of a building)

Parameter	Description
Space Name	Specifies the space name; the spaces in an adjacency matrix are listed by space name, and the space name can also be used in space labels. Select a name from the list, or select Custom and enter a name in the field below the pull-down list. The top portion of the list contains the space names that are currently used in the drawing. The lower portion of the list displays all available space names that are still unused, compiled from lists in the Vectorworks program folder, your user folder, and your designated workgroup folder. To customize a space name list, select Edit List from the bottom of the list to open the Edit Space - Space Name List dialog box. See “Editing Lists of Space Names and Occupant Organizations” on page 420. 
Occup. Organization	Specifies the organization associated with the occupant; the spaces in the stacking diagram are grouped by the organization name. This field is very similar to the **Space Name** field above. The list is separated into sections based on whether the available names are currently in use. Select a name from the list, or select Custom and enter a name in the field below the pull-down list. To customize an occupant organization list, select Edit List from the bottom of the list to open the Edit Space - Occ Organization List dialog box.
Occupant Name	Specifies the individual occupant name
Proposed Area	Specifies the programmed area for the space

Parameter	Description
Assign Zones	Click the column to the left of a zone name to select it. Up to five zones of different types can be assigned to a space; only one zone can be assigned from each zone type (HVAC, Preservation, Security, and so on). When you edit a space from the Object Info palette, click the **Assign Zones** button to open a dialog box with the same zone information that displays on the Occupancy pane. To customize the zones list, click **New** to open the Create Zone dialog box. • To add a zone to an existing zone type, select the **Zone Type**, enter the **Zone Name**, and click **OK**. • To create a new zone type, select Add New Zone Type from the **Zone Type** list, enter a name, and click **OK**. To delete a zone, click the zone name, and click **Delete** to remove the zone from the list. The **IFC Zones** command attaches IFC data to space zones. See "Assigning IFC Data to Space Zones" on page 1748.
GSA Occupancy	If **Enable GSA** is selected (Advanced Settings pane), opens the GSA Occupancy dialog box to enter the data required for GSA projects (see "GSA Data" on page 429). When the drawing is complete, select **File > Export > Export IFC Project** to save the file in IFC format.

Space Settings: 2D Boundaries & Area Pane

Many parameters on this pane are available only if the space is connected to adjacent walls.

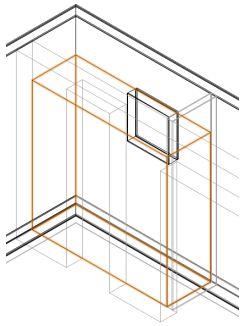
Click to show/hide the parameters.

Parameter	Description
Boundary Display	
Show 2D Boundary	Displays the polyline that represents the space; if this option is selected, also select an option for the **2D Boundary Display**.
2D Boundary Display	Specifies what to display for the space: Inside Walls less Columns, Inside Wall Faces, Wall Centerlines, Net Boundary, or Gross Boundary. A Net boundary is defined by the inside face of the surrounding walls; the Gross boundary display is defined by the **Gross Boundary Def.** setting. If the area and perimeter values for the space are used in a worksheet, the values always match the displayed boundary.

Parameter	Description
Gross Net Boundary	
Gross Boundary Def.	Specifies the definition to use for the gross boundary: Wall Centerlines, Building Gross, Outer Wall Core, Center Wall Core, BOMA Rentable, or Custom. If Outer Wall Core or Center Wall Core is selected but the wall has no core component, the space boundary is defined by the outer edge or center, respectively, of the entire wall. The Custom option is only available for an existing space object surrounded by walls. If Custom is selected, click **Edit Gross Custom** to open the Edit Custom Gross Boundary dialog box. Do one of the following: • Select each wall in the display window, and then select the appropriate bounding definition for it. • Click **Reset Gross Boundary** to set all boundaries for the space to one of five pre-set options (Centerlines of All Walls; Centerlines of Interior Walls, Outer Core Component of Exterior Walls; Centerlines of Interior Walls, Center Core Component of Exterior Walls; Centerlines of Interior Walls, Inside Face of Exterior Walls; Centerlines of Interior Walls, Outside Face of Exterior Walls).
Net Boundary Definition	Specifies the definition to use for the net boundary: Inside of Walls, or Custom. If Custom is selected, click **Edit Net Custom** to open the Edit Custom Net Boundary dialog box: Enter an amount to offset the space boundary from the wall. When the space is edited from the Object Info palette and it contains wall projections, columns, and/or pilasters, also choose whether to exclude wall projections and column “islands” from the area calculation.
Area Calculation	
Net Area Modifier	Adjusts the net area value by the specified percentage
Gross Area Modifier	Adjusts the gross area value by the specified percentage
Calculate Room Dimensions	Automatically calculates the **Length** and **Width** dimensions of the space object; when deselected, the **Length** and **Width** can be entered manually
Length/Width	Displays the dimensions of the space; when **Calculate Room Dimensions** is deselected, the **Length** and **Width** can be entered manually. Manual changes affect the display of the length and width values, but do not change the actual dimensions of the space object.

Space Settings: 3D Boundaries Pane

Click to show/hide the parameters.

Parameter	Description
Show 3D	Select to display the space volume boundaries in 3D 
Volume Display	Select whether to display the Net or Gross volume; the actual net or gross volume depends on the height boundary selections and any offsets
Height Net / Gross Height	
Height	Directly sets the desired net or gross height of the space. When the space height is determined manually by this method, the **Top Bound** property of the space is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the space is bound by the layer wall height value or by a story level, the space height displays automatically.
Top Bound	Sets the vertical reference that determines the top of the space. The **Layer Wall Height** value is set by the design layer (see “Setting Design Layer Properties” on page 165). Alternatively, the top of the space can be bound by one of the story levels defined for the story or the story above it. By setting the top of the space to a level type, if the elevation of the associated story changes, the height of the space changes automatically to match.
Top Offset	Sets the offset of the top of the space from its specified top bound height
Bottom Bound	Sets the vertical reference that determines the bottom of the space. Alternatively, the bottom of the space can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the space to a level type, if the elevation of the associated story changes, the height of the space changes automatically to match.
Bottom Offset	For the bottom of the space, sets the offset from its specified bottom bound height

Space Settings: Space Label 1, 2, and 3 Panes

As many as three space labels can be independently formatted and positioned for each space. The Space Label 1 pane is available by default; to add the Space Label 2 and Space Label 3 panes to the dialog box, select **Enable multiple labels** on the Advanced Settings pane (see “Space Settings: Advanced Settings Pane” on page 416).

Click to show/hide the parameters.

Parameter	Description
Display Space Label	Select to display the space label on the drawing Each label that is displayed is added to the Space Label Preview column on the right side of the dialog box.
Class	Select a class for the entire space label, or select New to create a new class (see "Creating Classes" on page 177). Alternatively, when you set the default preferences for the **Space** tool, select <Space Class> which places the label in the same class as the space object. Individual elements of the space label may be assigned to different classes, as well. To control the visibility of each space label separately, assign each label to a different class.
Space Label Symbol	Select the symbol to use for the space label from the list of symbols available in this document. To create a new text-based symbol, or to designate another symbol within the document as a space label, click **Manage Space Labels**.
Space Label Angle	Enter an angle to rotate the label, if desired
Fields list	The fields defined for the label symbol display; when a field is selected from the list, the field is highlighted in red in the Space Label Preview pane. To edit a field, select it from the list, and then do the following: • To use a different piece of data for a label (for example, to show the gross area instead of the net area), select the new definition from the **Format Field** list. From the Space Label pane, you can only edit the fields currently on the list. To add or delete fields, you must edit the label symbol. • To change the prefix or suffix that will display along with the field value on the label, click **Edit** to open the Edit Format dialog box. Enter the desired values and click **OK**.
Manage Space Labels	Opens the Manage Space Labels dialog box to create, rename, duplicate, or delete the space label symbols for this document; see "Customizing Space Labels" on page 421 for details
Save As Default	Opens a dialog box to save the current space label settings to a template file in your user folder, which makes it one of the available labels for new documents. Enter a name for the label and click **OK**. The template file is created (or updated) automatically in the space stamp file in the user folder's default library (see "Resource Libraries" on page 219).

Space Settings: Leader Line Pane

Parameters to position the leader line for space label 1 are available by default; leader line parameters for space labels 2 and 3 are available only if **Enable multiple labels** is selected on the Advanced Settings pane. The leader lines' class and appearance are set on the 2D Attributes pane (see "Space Settings: 2D Attributes Pane" on page 414).

Click to show/hide the parameters.

Parameter	Description
Display Leader Line 1/2/3	Select to display a leader line for the corresponding space label
Start Point on Space	Specifies whether to start the leader line on the space's anchor point or edge
End Point on Label	Specifies where to end the leader line relative to the space label. Auto position: Automatically centers the leader line end point on the closest center point of the space label's bounding box. Custom: Adds a control point at the end of the leader line; the control point can be moved to define where the leader line ends. This option is available only if an existing space is being edited, not as a default setting for new spaces. Nearest Locus: Ends the leader line on the space label locus point nearest to the space's anchor point. This option is available only if the chosen space label symbol contains one or more loci.

Space Settings: 2D Attributes Pane

These settings apply to the space object only; the space label is a symbol, which can be edited with the **Edit Layout** button, or from the Resource Browser (see "Editing Existing Space Labels" on page 422).

Click to show/hide the parameters.

Parameter	Description
Space Object Class	Select a class for the entire space object, or select New to create a new class.
2D Boundary	When you set the default preferences for the **Space** tool, only the **Class** parameter is available for the 2D Boundary, because the fill and pen attributes are determined by the Attributes palette. When you edit a space using the **Settings** button from the Object Info palette, all of the 2D boundary parameters are available.
Class	Select a class for the 2D boundary, or select New to create a new class. Alternatively, select <Space Class>, which places the boundary in the same class as the space object.
Fill Style	Select a fill style for the space object, and then customize the fill using the additional fields that display. For example, select the color for a solid fill, or select a hatch from the list of those available. For hatch, tile, gradient, and image fills, click **Edit** to open a dialog box for editing the fill. To use the fill specified for the selected class, select Class Style.
Pen	Select the pen style for the space object, and then customize the pen using the additional fields that display. For example, select the thickness for a solid pen, or select a color for a dash pen. To use the pen specified for the selected class, select Class Style.
Opacity	Use the slider or enter a percentage to specify the opacity of the space object, or click **Use Class Opacity** to use the opacity setting for the selected class.
Leader Line	If the label is placed outside the space boundary, a leader line is drawn automatically
Class	Select a class for the leader line, or select New to create a new class. Alternatively, select <Space Class>, which places the leader line in the same class as the space object.

Parameter	Description
Pen	Select the pen style for the leader line, and then customize the pen using the additional fields that display. For example, select the thickness for a solid pen, or select a color for a dash pen. To use the pen specified for the selected class, select Class Style.
Marker Start/End	To include start or end markers for the leader line, check the appropriate box(es). Then select the style for the markers, or select Custom to create a new marker style.

Space Settings: 3D Attributes Pane

Click to show/hide the parameters.

Parameter	Description
Class	Select a class for the 3D attributes of the space object, or select New to create a new class. Alternatively, select <Space Class>, which places the 3D attributes in the same class as the space object.
Fill Style	Select a fill style for the space object, and then customize the fill using the additional fields that display. For example, select the color for a solid fill, or select a hatch from the list of those available. For hatch, tile, gradient, and image fills, click **Edit** to open a dialog box for editing the fill. To use the fill specified for the selected class, select Class Style.
Pen	Select the pen style for the space object, and then customize the pen using the additional fields that display. For example, select the thickness for a solid pen, or select a color for a dash pen. To use the pen specified for the selected class, select Class Style.
Texture	To use a texture for the space object, select a texture from the list of those available. To use the texture specified for the selected class, select Class Style.

Space Settings: Room Finishes Pane

Click to show/hide the parameters.

Parameter	Description
On Room Finish Schedule	Includes the space object in the Room Finish Schedule; if enabled, select the appropriate finishes for the ceiling, walls, base trim, and floor of the space object. Alternatively, to assign room finishes to multiple existing spaces at the same time, select the desired spaces and click **Assign Room Finish** in the Object Info palette. This opens the Assign Room Finishes dialog box, which has the same functionality as the Room Finishes pane.
Ceiling	Select a finish for the space object's ceiling. The top portion of the list contains the ceiling finishes that are currently used in the drawing. The lower portion of the list displays all available ceiling finishes that are still unused, compiled from lists in the Vectorworks program folder, your user folder, and your designated workgroup folder.
North, East, South, and West Walls	Select a finish for each of the space object's walls. The top portion of the list contains the wall finishes that are currently used in the drawing. The lower portion of the list displays all available wall finishes that are still unused, compiled from lists in the Vectorworks program folder, your user folder, and your designated workgroup folder.

Parameter	Description
Base Trim	Select a finish for the space object's base trim. The top portion of the list contains the base trim finishes that are currently used in the drawing. The lower portion of the list displays all available base trim finishes that are still unused, compiled from lists in the Vectorworks program folder, your user folder, and your designated workgroup folder.
Floor	Select a finish for the space object's floor. The top portion of the list contains the floor finishes that are currently used in the drawing. The lower portion of the list displays all available floor finishes that are still unused, compiled from lists in the Vectorworks program folder, your user folder, and your designated workgroup folder.
Remarks	Allows entry of additional remarks about the finishes
Edit Finishes	Opens the Edit Room Finishes dialog box

Once the room finish information is entered, add the Room Finish Schedule to the drawing file.

Space Settings: Additional Data Pane

Click to show/hide the parameters.

Parameter	Description
Fields list	You can attach data to the space object and display the information in a space label, or in a worksheet. All available fields display in a list, along with any data that has been entered for the fields. For most fields, simply click the field name in the list, and then enter the desired data in the entry field that displays below the list of field names. To attach data that is not available on the list, you can use up to ten Additional Info fields; click **Rename User Fields** to name these fields. The Room ID and Formula fields have additional functionality. The Room ID allows you to specify an identifier based on a formula that can include data fields from the space as well as user-defined static text. This ID can then be used as a variable in a space tag symbol. For example, you might define a space's Room ID using the design layer and space number fields as variables.
Rename User Fields	If you entered data for any of the Additional Info fields, click this button to open the Rename User Fields dialog box and enter names for the fields. When you click **OK**, a message displays that the change will be applied to all space objects that use that field.
Use IFC Data	When the file is saved in IFC format, includes the IFC data for the space object
IFC Data	If **Use IFC Data** is selected, opens the IFC Data dialog box to enter the data for various space properties (see "Viewing and Editing IFC Data" on page 1748). When the drawing is complete, select **File > Export > Export IFC Project** to save the file in IFC format.

Space Settings: Advanced Settings Pane

Click to show/hide the parameters.

Parameter	Description
Enable GSA	Includes the GSA data for the space object when the file is saved in IFC format; click **GSA Occupancy** (on the Occupancy pane) to assign the GSA Occupancy information for the space. See “GSA Data” on page 429.
Enable multiple labels	Adds Space Label 2 and Space Label 3 panes to the Space Settings dialog box; see “Space Settings: Space Label 1, 2, and 3 Panes” on page 412. When this option is selected, parameters for leader lines 2 and 3 are also added to the Leader Line pane.
Make picked up attributes the default	When the Pick Up Attributes mode of the **Space** tool is used, the attributes selected in the Eyedropper Transfer Properties list are set as the default for new spaces. If the 2D Boundary Attributes are selected, the picked up attributes will be set as the default in the Attributes palette. This allows you to create new spaces with the same attributes as an existing space.
Eyedropper Transfer Properties	Specifies which properties of a space object will be transferred when the Pick Up Attributes and Apply Attributes modes of the **Space** tool are used. The available attributes display in the **Attribute** column, grouped by category. To see the attributes in a category, click the disclosure arrow to the left of the attribute name. Click the **Use** column next to an item to select it; if the item is a category heading, such as Finishes, all items in the category are selected.
Space Properties on Object Info Palette	Specifies which properties of a space object will be displayed for editing on the Object Info palette. The available properties display in the **Property** column, grouped by category. To see the properties in a category, click the disclosure arrow to the left of the property name. Click the **Show** column next to an item to select it; if the item is a category heading, such as Occupancy, all items in the category are selected.
Restore Default Properties	Resets the space properties on the Object Info palette to the default settings

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Space Properties
Using Auto-Numbering
Editing Lists of Space Names and Occupant Organizations
Customizing Space Labels
Room Finishes
Customizing the Space Settings
GSA Data
Creating Spaces with the Space Tool
Editing Space Boundaries

Space Properties

The space parameters are described in “Space Settings” on page 406. Only the parameters that are different are described here.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Assign Zones (Occupancy section)	Opens the Assign Zones dialog box, which has the same functionality as the Assign Zones section of the Occupancy pane of the Space Settings dialog box
Auto-Boundary	Specifies whether the space object is associated with a set of walls, or whether its boundaries were created manually. For an auto-bounded space, if the walls are edited (whether moved, reshaped, or replaced, or due to a change in the wall style, the wall thickness, or the wall components), the space edge path updates automatically.
Update Boundary (2D Boundaries & Area section)	Available only if **Auto-Boundary** is enabled. Updates the boundary of the space to match the surrounding walls after visible walls are added, deleted, or moved; also updates the boundary if a custom space boundary definition is changed (for example, if the space boundary is offset from the walls).
Assign Room Finish (Room Finish section)	Available only if **On Room Finish Schedule** is enabled; opens the Assign Room Finishes dialog box, which has the same functionality as the Room Finishes pane of the Space Settings dialog box. This option is available when multiple spaces are selected, so the same finishes can easily be assigned to all selected space objects.
Matrix Order (Additional Data section)	Specifies the location of the space object in the adjacency matrix

Using Auto-Numbering

The Vectorworks Architect program can automatically number spaces as they are created, or you can enter numbers manually. If you use the auto-numbering feature, the Numbering pane of the Space Settings dialog box has additional features to customize numbers and to adjust numbers when needed.

Custom Number Styles

You can create custom space numbers that contain information other than just a counter, such as the floor the space is on, or what zone the space is in.

To create, edit, or delete a number style:

1. Select the **Space** tool, and then click Preferences on the Tool bar.
2. On the Space Settings dialog box, select Numbering from the left pane to display the Numbering pane.
3. Click **Edit Number Styles** to add, edit, or delete the available number styles.

   The Edit Number Style dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Space Number Style	Enter a descriptive name for the number style
Prefix / Suffix	Specifies additional information to display before and/or after the space number. If a prefix or suffix is desired, enter text and/or select one or more items from the list of variables to the right of the fields to insert automatically generated data. For example, your number style might have the floor the space is on, followed by a period and the automatically assigned space number. In the **Prefix** field, select the Floor variable, and then type a period; the space number variable is added to the definition automatically.
Style Name/Style Definition/Saved in	Displays the parameters of number styles that exist in the document. Select an item from the list to edit it.

4. Edit the number styles as follows:
   - To add a new number style, type a style name into **Space Number Style**, and add a **Prefix** and/or **Suffix** if desired. Click **Save**; the number style is added to the **Style Name/Style Definition/Saved in** list.
   - To edit a number style, select it from the **Style Name/Style Definition/Saved in** list and then edit the **Prefix** and/or **Suffix** as needed. Click **Save** to save the changes.
   - To delete a number style, select it from the list and then click **Delete**.
5. When all editing is complete, click **OK** to close the Edit Number Style dialog box.

Validating Auto-Numbering

Occasionally the auto-numbering may need adjustment. For example, if a space is deleted after creation, one of the numbers will be missing from the sequence. If a space was duplicated, its number may be used twice for the same label type. The Vectorworks Architect program can help you detect and correct such problems.

To validate auto-numbering:

1. Select the space object.
2. From the Object Info palette, click the **Settings** button to open the Space Settings dialog box, and then select the Numbering pane.
3. Click **Validate Auto-Numbering**.

   The Validate Auto-Numbering dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Total Counter of this Tag / Start Value / Increment	For the selected space object, displays the total number of space objects that use the same label, along with the start value and increment set for auto-numbering
Ctr / Space Number	Each space object that uses the same label as this space displays in a list, which shows the space's current counter (Ctr) and Space Number. The list also includes an entry if a number in the sequence is missing.
Correction button (name varies)	When a list item with a numbering error is selected, a button to correct the problem displays below the list.

4. If there is a message next to a space number, select it and then click the correction button that displays beneath the list.

Message	Problem	Correction
missing	A number in the sequence is missing	Click **Close Gaps** to renumber all spaces that fall after the gap in the sequence
Value invalid (< Start)	The space's current number is less than the start value	Click **Assign valid value** to automatically renumber the spaces as needed
(multiple)	Multiple spaces have this number	Click **Increase by Increment** to automatically renumber the space

5. When all problems have been corrected, click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Space Settings
Creating Spaces with the Space Tool

A Editing Lists of Space Names and Occupant Organizations

The Occupancy pane of the Space Settings dialog box has **Space Name** and **Occup. Organization** fields from which you can select values to assign to spaces. The names are collected from files in the Vectorworks application folder, your user folder, and any folder designated as your workgroup folder.

From the Vectorworks application folder, the default content from the standard occupant organization and space name libraries is always available. These lists are called "BOMA Standard Occupant Organizations.txt" and "Generic Space Names.txt." You cannot modify these lists, but you can import and export items from them. Use the Space Settings dialog box to create custom versions of these lists, or to create new lists, as needed. Edits are automatically saved to your user folder.

In a workgroup environment, create space name and occupant organization lists and place them in a shared location. The folder structure must be the same as that of the default content libraries. See "Customizing the Space Settings" on page 428.

To edit a list of space names or occupant organizations:

1. Select the **Space** tool, and then click Preferences on the Tool bar.

   Alternatively, select an existing space object, and from the Object Info palette, click the **Settings** button.
2. On the Space Settings dialog box, select Occupancy from the left pane to display the Occupancy pane.
3. Click **Space Name** or **Occup. Organization**, and select Edit List from the bottom of the list.

   The appropriate editing dialog box opens.
4. Select the list to view.
   - To view items from the list in the current document, click **Active Document**.
   - To view items from a list external to the drawing file, click **Default Folder**, and then select from the **List** options. Lists from the Vectorworks application folder, your user folder, and your designated workgroup folder display.
5. Edit, import, or export items as described in the following table. The options available depend on what type of list you are editing.

Edit Function	Applicable Files	Procedure
Rename the list	User lists	Click **Rename**, enter the new list name, and click **OK**. The list name in the edit dialog box changes.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Edit Function	Applicable Files	Procedure
Change the list order	Active document, user lists	Click the item number and drag it up or down to the desired position.
Add an item	Active document, user lists	Click **New**. • If the active document list or a user list is selected, a blank item is added to the current list. Enter a name in the **Item Value** field. • If a list from the application folder or a workgroup folder is selected, an alert displays, and you are prompted to add the item to a list in your user folder instead. Answer **Yes** to open a dialog box to add an item. Enter a name in the **Item Value** field. To create a new file, select **Save to New File** and enter a unique name. To add to an existing file, select **Add to Current File** and select a file from the pull-down list. Click **OK** to display the new item and list.
Edit an item name	Active document, user lists	Select the item and enter the new name in the **Item Value** field.
Delete an item	Active document, user lists	Select the item and click **Delete**. The item is removed from the list, and the items below it on the list are renumbered.
Import items	User, workgroup, and application lists	Select one or more items and click **Import**. The items are added to the Active Document list.
Export items	Active document, user, workgroup, and application lists	Select one or more items and click **Export** to open the export dialog box. To create a new file, select **Export to New File** and enter a unique name. To add to an existing file, select **Add to Current File** and select a file from the pull-down list. Click **OK** to complete the export.

6. Click **OK** to save the changes and close the edit dialog box. If you changed an external list, a message displays the path in your user folder where the edited file was saved; click **OK**.
7. Click **OK** to close the Space Settings dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
Customizing the Space Settings
Space Settings
Creating Spaces with the Space Tool

Customizing Space Labels

The Vectorworks Architect product provides several space labels, which you can use or modify as needed. You can also create new custom labels, import a space label symbol from another document, or convert a symbol in the current document into a space label.

Custom space labels are symbols that are designated as space labels from the Space Settings dialog box. Like any other symbol, by default a custom label is only available in the document in which it is created. To make a custom space label available from the Space Settings dialog box for all of your documents, use the **Save as Default** option to save the label
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

to a template file. The Space Default template file is created in your user folder automatically, in [User Folder]\Libraries\Defaults\.

To share custom space labels with a workgroup, create a template file that contains the labels (see “Creating Space Labels” on page 423). Place the file in the workgroup’s default labels folder. The labels then display on the list of available symbols on the Space Label pane of the Space Settings dialog box. See “Customizing the Space Settings” on page 428.

Click here for a video tip about this topic (Internet access required).

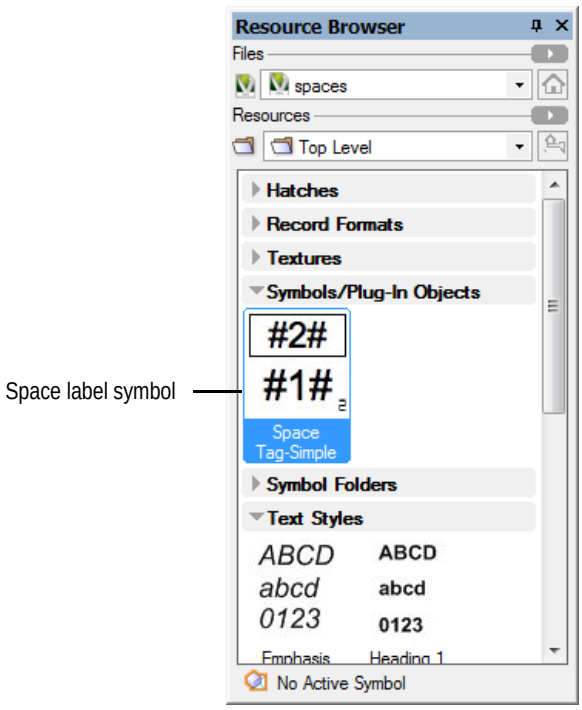
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Existing Space Labels

You can easily make minor adjustments to an existing space label. For example, you might adjust the font style or color, or change the graphics that are included in the symbol.

To modify an existing space label symbol:

1. Do one of the following:
   - From the Space Label pane of the Space Settings dialog box, select the label to be edited; a preview of it displays on the right side of the dialog box. Click **Edit Layout** to open the symbol editing window.
   - If the label displays in the Symbols/Plug-in Objects group in the Resource Browser, select it and then select **Resources > Edit**. From the Edit Symbol dialog box, select the **2D Component** option, and click **Edit** to open the symbol editing window.



2. In the symbol editing window, edit the text formatting, graphic components, and attributes of the symbol as needed. Note that a space label is a page-based symbol, which means that its size in the editing window is the same as it will be when inserted into the drawing.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. When the changes are complete, click **Exit Symbol**.
4. The symbol definition and all instances of the label in the drawing are updated automatically.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating New Symbols
Creating Space Labels

A Creating Space Labels

If none of the available space labels are suitable, create a new label.

To create a new space label:

1. From the Space Planning tool set or the Building Shell tool set, select the **Space** tool.
2. From the Tool bar, click Preferences to open the Space Settings dialog box.
3. Select the desired space label pane, and click **Manage Space Labels**. The Manage Space Labels dialog box opens.

Item	Description
Symbols list	The left side of the dialog box displays a list of all space label symbols currently available, along with the current status of each: • "Default" symbols are those available from default content, but not yet used in the drawing; once a default space label symbol is used, it is imported into the drawing file's resources and the status becomes Specified. Default symbols cannot be renamed or deleted. • "Specified" symbols are currently used in the drawing. • "Unspecified" symbols are currently in the drawing file's resources, but not yet used or saved as a default.
Symbol preview	The top right side of the dialog box displays a preview of the selected symbol; if a label field has not been defined yet, a red "Not specified" message displays on the image
New Text Tag	Opens the New Text Tag dialog box, to create a new space symbol; proceed with step 4
New Symbol Tag	Opens the New Symbol Tag dialog box, to designate a symbol in the current document as a space symbol.
Rename	Opens the Rename Label dialog box; enter a new name for the symbol and click **OK**. If the selected symbol has a "Default" status, a message displays that it cannot be renamed.
Duplicate	Opens the Duplicate Label Symbol dialog box; enter a name for the duplicate symbol and click **OK**.
Delete	Opens the Delete Space Label dialog box. Specify whether to delete the space symbol from the symbol library entirely, or to keep the symbol, but remove the space label definition from it (this removes the label from the list of current space label choices). Click **OK** to delete the label. If the selected symbol has a "Default" status, a message displays that it cannot be deleted.

4. Click **New Text Tag**; the New Text Tag dialog box opens to the Data Fields tab.
5. All available data fields for spaces display in the list on the left. Select the fields to be used in the symbol, and click **Add** (or click **Add All**) to move them to the list on the right. To remove fields from the symbol list, select them and click **Remove** (or click **Remove All**). Alternatively, double-click a field name to move it from one list to the other.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

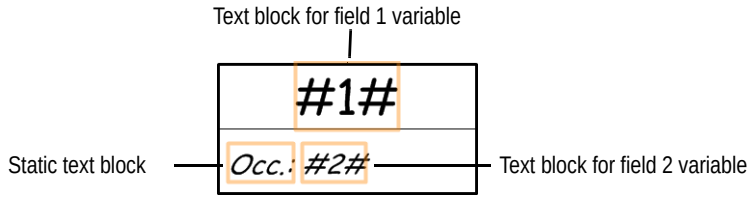
6. In the list of selected symbol fields on the right, each field is assigned a sequential number, which corresponds to a variable in the symbol layout (for example, #2#). To change a field's number, click its number column, and drag the field up or down to the desired location on the list.
7. Select the Attributes tab to specify how all fields in the symbol will display.

Item	Description
Font	Select a font for the field text from the list of fonts available on the system
Size	Select a font size from the list of standard sizes available in points. To enter a custom point size, or to enter a size in different units, select **Set Size** to open the Set Text Size dialog box. Enter a **Size**, and then select the appropriate **Unit**: Points, Page Millimeters, or Page Inches.
Style	Select a style for the field text from the list of styles available. On Mac, select **Custom Style** to open the Custom Style dialog box, to specify a different combination of styles.
Alignment	Select a type of alignment for the field text: Left, Center, Right, or Standard (justify)
Fill	Select a fill for the background of the field text objects: None, Pattern, or Solid. Alternatively, to assign the fill by text object class, select Class Style.
Color	If a Solid fill was selected, select the fill color
Pattern/Fore/Back	If a Pattern fill was selected, select the Pattern, and the foreground and background colors
Text Color	Select a color for the field text

8. Click **OK** to save the changes and return to the Manage Space Labels dialog box. Click **OK** again to return to the applicable space label pane.
9. Complete the definition of the label.
10. To save the label to your template, select the **Save as Default** option.
11. To change the appearance of the label (for example, to rearrange the data fields or to add graphic elements), click **Edit Layout** to open the symbol editing window.

    The symbol is a group object, in which each defined field is a text block that contains text in the following format: #(natural number)#. You may ungroup the blocks and move them or change their attributes, but do not change the contents of the text blocks.

    In the following example, the top part of the symbol is a center-aligned text block that contains a variable for field 1; the **Space Number** will be assigned to this field. The bottom part of the label has a text block that contains a variable for field 2; the **Occupant Name** field will be assigned to this field. Another text block was added, which serves as a label for the occupant name. A rectangle and a line were also added to enhance the label.



12. When the edits are complete, click **Exit Symbol**. The new label is now available to assign to spaces for this document.

Customizing Space Labels
Creating New Symbols
Converting Symbols into Space Labels

Converting Symbols into Space Labels

If you already have a symbol you want to use, you can convert it for use as a space label.

To convert a symbol into a space label:

1. Ensure that the symbol includes a text block for each piece of space object data you want to display on the label. The text in each field block should be in the following format: #(natural number)#. The example below has four fields. Apply the font, size, style, alignment, and fill and pen style desired to each text block.

   Also, ensure that the space label symbol is specified as having world-based units (not page-based).

2. From the Space Planning tool set or the Building Shell tool set, select the **Space** tool.
3. From the Tool bar, click Preferences to open the Space Settings dialog box.
4. Select the desired space label pane, and click **Manage Space Labels**. The Manage Space Labels dialog box opens.
5. Click **New Symbol Tag**; the New Symbol Tag dialog box opens.
6. The available symbols and symbol folders in the current document display; symbols already defined as space labels do not display. Locate and select the symbol to be used for the new label, and click **OK**.
7. On the Manage Space Labels dialog box, notice that the preview window shows that the fields are not specified yet. Click **OK** to return to the Space Settings dialog box.
8. Complete the definition of the data fields. The new label is now available to assign to spaces for this document.
9. To save the label to your template, select the **Save as Default** option.

Space Settings: Space Label 1, 2, and 3 Panes
Customizing Space Labels

Room Finishes

Creating, Editing, and Deleting Room Finishes

The Room Finishes pane of the Space Settings dialog box has **Ceiling, Base Trim, Floor,** and **North/South/East/West Wall** fields from which you can select values to assign to spaces. The names are collected from files in the Vectorworks application folder, your user folder, and any folder designated as your workgroup folder.

From the Vectorworks application folder, the default content from the room finish libraries is always available. You cannot modify these lists, but you can import and export items from them. Use the Space Settings dialog box to create custom versions of these room finish lists, or to create new lists, as needed. Edits are automatically saved to your user folder.

In a workgroup environment, create room finish lists and place them in a shared location. The folder structure must be the same as that of the default content libraries. To edit a room finish list:

1. Select the **Space** tool, and then click Preferences on the Tool bar.
2. On the Space Settings dialog box, select Room Finishes from the left pane to display the Room Finishes pane.

   Alternatively, select one or more existing space objects and click **Assign Room Finish** in the Object Info palette to open the Assign Room Finishes dialog box. (If the button is not enabled, select **On Room Finish Schedule**.) If room finishes are different for multiple selected space objects, the options display as a blank. Any edits to that parameter affect all of the selected spaces.
3. Click **Edit Finishes** to add, edit, or delete the available room finishes.

   The Edit Room Finishes dialog box opens.
4. Select the list of room finishes to view.
   - To view items from the current document, click **Document Finishes**, and then select from the **Location** options (ceiling, walls, baseboard, or floor).
   - To view items from a list external to the drawing file, click **Default Finishes**, and then select from the **Location** options. Lists from the Vectorworks application folder, your user folder, and your designated workgroup folder display.
5. Edit, import, or export items as described in the following table. The options available depend on what type of list you are editing.

Edit Function	Applicable Files	Procedure
Rename the list	User lists	Click **Rename**, enter the new list name, and click **OK**. The list name in the edit dialog box changes.
Change the list order	Active document, user lists	Click the item number and drag it up or down to the desired position.

Edit Function	Applicable Files	Procedure
Add an item	Active document, user lists	Click **New**. • If an active document list or user list is selected, a blank item is added to the current list. Enter a **Key** and **Description**. • If a list from the application folder or a workgroup folder is selected, an alert displays, and you are prompted to add the item to a list in your user folder instead. Answer **Yes** to open a dialog box to add an item. Enter a **Key** and **Description**. To create a new file, select **Save to New File** and enter a unique name. To add to an existing file, select **Add to Current File** and select a file from the pull-down list. Click **OK** to display the new item and list.
Edit an item **Key** or **Description**	Active document, user lists	Select the item and enter the new value in the appropriate field. The **Key** is the ID that displays for the selected finish in the space label (if applicable), Room Finish Legend, and Room Finish Schedule. **Description** text displays for the selected finish in the Room Finish Legend.
Delete an item	Active document, user lists	Select the item and click **Delete**. The item is removed from the list, and the items below it on the list are renumbered.
Import items	User, workgroup, and application lists	Select one or more items and click **Import**. The items are added to the appropriate list in the current document.
Export items	Active document, user, workgroup, and application lists	Select one or more items and click **Export** to open the export dialog box. To create a new file, select **Export to New File** and enter a unique name. To add to an existing file, select **Add to Current File** and select a file from the pull-down list. Click **OK** to complete the export.

6. Click **OK** to save the changes and close the edit dialog box. If you changed an external finish list, a message displays the path in your user folder where the edited file was saved; click **OK**.
7. Click **OK** to close the Space Settings dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
Customizing the Space Settings
Space Settings
Creating Spaces with the Space Tool

Assigning a Room Finish

Room finishes can be applied to space objects (see “Creating Spaces from Existing Walls” on page 405). The information displays in the Room Finish Schedule and in the Room Finish Legend (see “Records and Schedules” on page 1859).

To assign finishes to a space object:

1. Select an existing space object.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. In the Object Info palette, select **On Room Finish Schedule** to add the finish information to the Room Finish Schedule. Click **Assign Room Finish**.

   The Assign Room Finishes dialog box opens.

   Alternatively, click **Settings** to open the Space Settings dialog box, and select the Room Finishes pane.

Click to show/hide the parameters.

Parameter	Description
Ceiling	Lists all of the defined finish items for ceilings
North, East, South, and West Walls	Lists all of the defined finish items for each wall
Base	Lists all of the defined finish items for baseboards
Floor	Lists all of the defined finish items for floors
Finish Remarks	Allows entry of additional remarks about the finishes
Edit Finishes	Opens the Edit Room Finishes dialog box

3. Select the desired finishes for each part of the room and click **OK**. The information is associated with the selected space object.
4. The Room Finish Schedule can be added to the drawing from the **VA Create Schedule** command or the Resource Browser. From the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder. Drag the Room Finish Schedule worksheet to the drawing. The worksheet is populated with information from the objects in the current drawing.To add a legend for the Room Finish Schedule, use the **Create Rm Finish Legend** command.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
Creating, Editing, and Deleting Room Finishes
Creating a Room Finish Legend
Creating Schedules
Space Settings
Creating Spaces with the Space Tool

Customizing the Space Settings

When you edit the space name, occupant organization, and room finish lists, the changes are saved to text files in your user folder. In a workgroup environment, you can create a set of files that contain standard settings for your office and share them with coworkers over a network.

To access the shared content, another Vectorworks user simply designates that folder as a workgroup folder in his or her Vectorworks Preferences. Then, when the user creates spaces in a drawing, the space name, occupant organization, and room finish list items are collected from files in the Vectorworks program folder, the user folder, and any folder designated as a workgroup folder.

To create custom space settings:

1. In the **Space** tool's preferences, customize the space name, occupant organization, and room finish lists as needed. The appropriate files are created in the user data folder specified in your Vectorworks preferences (see "User Folders Preferences" on page 57).
2. To share the custom files with a workgroup, put the files on the network in a workgroup folder. When coworkers set up this workgroup folder in their Vectorworks preferences, they also have access to the files (see "Sharing Custom Content Using Workgroup Folders" on page 216). The folder structure must be the same as the default
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

content room finish, standard occupant organization, and space name libraries provided in [Vectorworks]\Libraries. For example, to add a custom space name library, place the file in your workgroup default space name folder: [Workgroup]\Libraries\Defaults\Space - Space Name.

3. To share custom space labels with a workgroup, save the labels to your template file (see “Creating Space Labels” on page 423). Place the file in the workgroup’s default space stamp folder.
4. Remove the custom files you created from your user folder if you intend to use the workgroup settings. Remember that the contents of the user folder have priority over the contents of the workgroup folder.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Space Settings: Occupancy Pane
Custom Number Styles
Editing Lists of Space Names and Occupant Organizations
Customizing Space Labels
Creating, Editing, and Deleting Room Finishes

A GSA Data

The General Services Administration in the United States has set deliverables standards that require that projects be delivered in IFC format. In addition, they require that certain additional GSA-specific data be included with spaces.

To specify GSA data for a space object:

1. Select an existing space object.
2. In the Object Info palette, click **Settings** to open the Space Settings dialog box.
3. Select the Advanced Settings pane, and then click **Enable GSA**.
4. Select the Occupancy pane, and then click **GSA Occupancy**.

   The GSA Occupancy dialog box opens.

Click to show/hide the parameters.

Parameter	Description
OmniClass	Select the federal agency classification for the space by its function; to see the subclasses beneath a heading, click the disclosure arrow to the left of the heading
Space Category	Select the GSA category for the space; the numeric code for the category displays to the left
Space Type	Select the GSA code that describes the usage of the space; a description of the code displays to the right
Occupant	Select the federal agency code and name for the organization that will occupy the space; to see the subclasses beneath a heading, click the disclosure arrow to the left of the heading

5. Enter the GSA data for the space object and click **OK**.
6. When the drawing is complete and all required GSA data has been entered, select **File > Export > Export IFC Project** to save the file in IFC format.

GSA Spatial Program BIM Requirements

A GSA spatial program BIM has some specific modeling requirements. Use the following guidelines to create an optimal IFC export from the Vectorworks program, that is compatible with all GSA guidelines and standards.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

General

- Reduce, eliminate, or deactivate (by hiding appropriate classes) the 3D components of objects that do not contribute to the spatial definition of the model. While the IFC data format does support a number of "non-spatial" or "non-structural" items, they are not required for the current GSA spatial program BIM.
- Ensure that all wall intersections are correctly configured, so that an automatically bound space object can properly detect room extents.
- Ensure that all space objects have the following settings in the Object Info palette:
  - **2D Boundary Display** (2D Boundaries & Area section) is set to "Net Boundary"
  - **2D Net Boundary Definition** (2D Boundaries & Area section) is set to "Inside Face of Walls"
  - **Show 3D** (3D Boundaries section) is enabled
  - The **Volume Display** of the 3D boundary is set to "Net," and the height is set to properly represent the full height of the space, either manually, or based on story-aware settings

Columns and Pilasters

The GSA requires a "net" space boundary that wraps around pilasters and free-standing concrete columns, excluding them from the area calculations. However, the space boundary should not wrap around steel columns, or columns embedded in a wall; these should be ignored.

To create a GSA BIM-compatible vertical support, use either the **Column** or **Pilaster** tool. A column is restricted to symmetrical square or oval plan geometry. A pilaster must be set to Architectural, or Structural and Architectural, to comply with GSA requirements. The space boundary will wrap around the extents of these structures.

Columns that are exposed steel wide flanges or H-shapes do not need to be excluded from the GSA BIM area. If a structural steel column is required, use extruded plan geometry inside a hybrid symbol to create the column; the column will be recognized as 3D geometry, but it will not be excluded from the GSA BIM area in the space object.

Shafts

Per GSA guidelines, a shaft that has a floor penetration greater than nine square feet should be designated as its own space.

If this shaft is bounded completely by walls, simply create a space object inside the shaft. If the shaft is not bounded by walls, use the **Wall** tool's <Virtual wall> style to create a boundary; once the shaft is surrounded with virtual walls, create spaces in the room and shaft.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

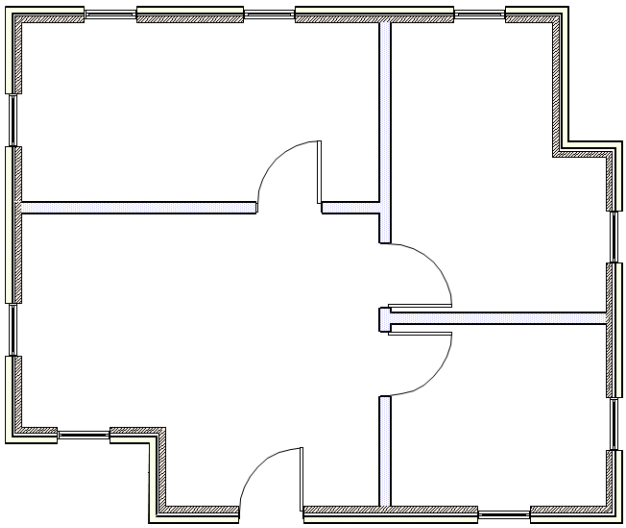
Space Settings
Creating Spaces with the Space Tool
Creating Columns and Pilasters

Creating Spaces from Walls

Floor plans can be created by first creating walls, and then automatically creating spaces to determine the area of each room or the gross area defined by the wall perimeter.

To create spaces from walls:

1. Create the walls and ensure that they are properly joined.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

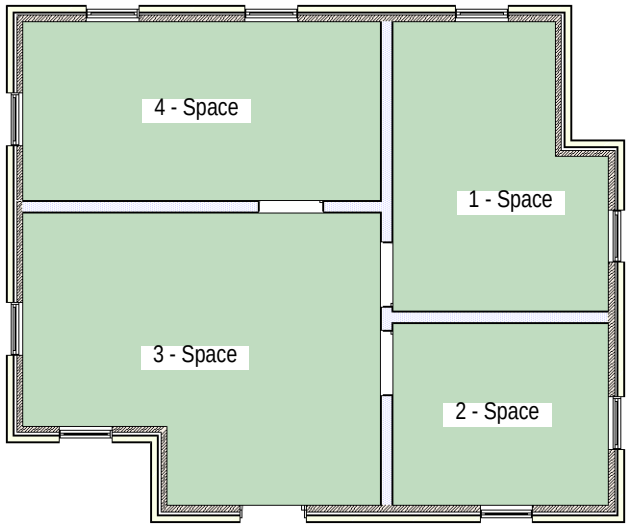


2. From the Attributes palette, set the 2D attributes for the spaces (fill, pen, opacity, and line thickness).
3. Select the **Space** tool, and from the Tool bar, select Preferences. Set the default parameters for the space objects.
4. Select **AEC > Space Planning > Create Spaces from Walls**. The Create Spaces from Walls dialog box opens. Specify how to create the spaces, the minimum area for creating spaces, and their location.

Click to show/hide the parameters.

Parameter	Description
Create Room Spaces	Creates space objects for each room defined by the walls in the source layer
Create Gross Floor Space	Creates a single space object equivalent to the entire outer boundary of the walls in the source layer
Source Layer	Specifies the layer that contains the walls
Destination Layer	Indicates the layer on which to create the spaces
Minimum area	Specifies the threshold for creating a space; spaces will be created only for areas above this value
Create Spaces from Selected Walls Only	Creates spaces only from selected walls; deselect to create spaces from all the walls in the source layer

5. Click **OK** to create the spaces on the destination layer.



Spaces are created for all areas enclosed by walls, or for the gross area defined by boundary of the walls. Default room name information is set and should be edited to complete the floor plan.

After the spaces are created, the original walls can still be modified. To update the spaces to conform to the new walls, select the **Create Spaces from Walls** command again. The existing spaces are reshaped without losing any space parameters that have already been specified.

If the original spaces were not drawn perfectly straight, select the **Create Walls from Spaces** command with the **Constrain Straight Walls** option. Now that the walls are straight, select the **Create Spaces from Walls** command to straighten the spaces.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

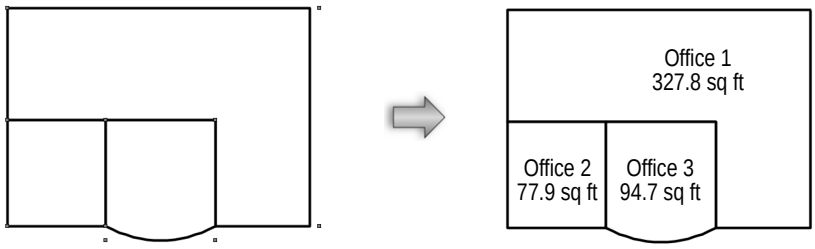
Space Settings
Creating Spaces with the Space Tool

Creating Spaces from Polylines

During the initial stages of a design, it may be easier to work with polylines instead of spaces. Use the **Modify > Add/Clip/Intersect/Combine into Surface** commands to modify the polylines to create the desired shapes. Once the polyline shapes are finalized, convert them into spaces to take advantage of the additional features that space objects provide, such as labels, area calculations, room finishes, and so on.

To convert polylines into spaces:

1. From the Attributes palette, set the 2D attributes for the spaces (fill, pen, opacity, and line thickness).
2. Select the **Space** tool, and from the Tool bar, select Preferences. Set the default parameters for the space objects.
3. Select the polylines to convert into spaces. This command also converts rectangles, rounded rectangles, polygons, arcs, circles, and ovals into spaces.
4. Select **AEC > Space Planning > Create Spaces from Polys**. The selected items are converted into spaces.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Another option is to draw a polyline and then select the **Create Objects from Shapes** command (see “Creating Polylines” on page 298 and “Creating a Stacking Diagram” on page 442).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Space Settings
Creating Spaces with the Space Tool

Getting Floorplan Information from a Solid Model

If a solid model has been used to develop a massing study, information can be extracted from the model to begin work on the floorplans. Specifically, gross area polylines can be created automatically for each building level directly from the solid model with the **Model to Floorplan** command. A stacking diagram can display the sum of the areas of these polylines, which makes it easy to determine the amount of floor area provided by a solid model before creating the floorplans. In addition, exterior walls can be created automatically. If the model changes after the polylines and walls are created, run the **Model to Floorplan** command again to update the polylines and walls.

To create a floorplan from a model:

1. Select the model. The solid model should be a CSG solid (addition, subtraction, union), solid primitive (sphere, hemisphere, cone, cylinder), extrude, or multiple extrude.

   This command cannot be used in Top/Plan view. Select the model from a 3D view.

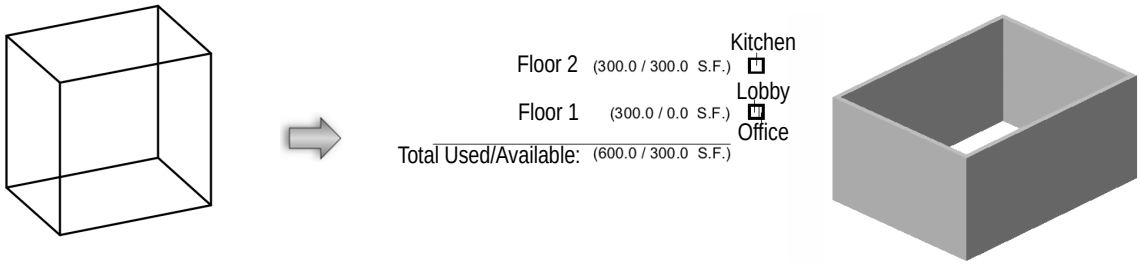
2. Select **AEC > Space Planning > Model to Floorplan**.

   The Model to Floorplan dialog box opens. Specify the layers to include and set the type of floorplan elements to create.

Click to show/hide the parameters.

Parameter	Description
Layer list	Lists the layers with heights that intersect the solid. Select the design layer(s) to include in the floorplan, and deselect layers, such as sheet layers or site model layers, which should not be included in the floorplan. A selected layer displays with a check mark.
Create Gross Area Polylines	For each selected layer, creates a polyline from the exterior perimeter of the model
Create Stacking Diagram	Creates a stacking diagram showing the available area on each layer (see “Creating a Stacking Diagram” on page 442)
Create walls	Creates walls based on the model perimeter
Position	Creates exterior walls based on the model perimeter, along the inside, outside, or center of the perimeter
Wall Style	Specify a wall style to use for the exterior walls from either the default content or the current file’s content

3. Click **OK** to create the floorplan objects. If selected, polylines, walls, and a stacking diagram are placed on the drawing. To update the floorplan objects based on model changes, select the **Model to Floorplan** command again.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Resource Libraries

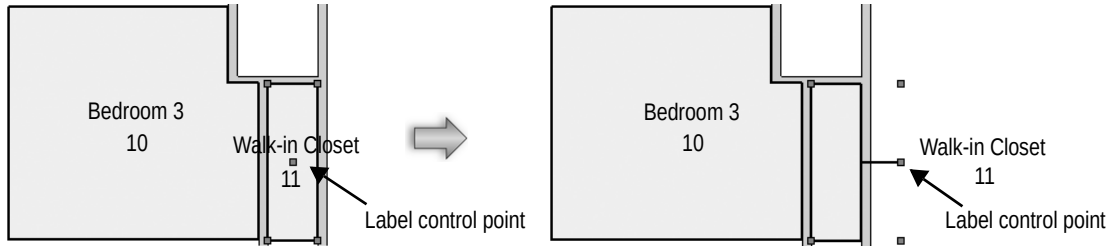
Editing Space Boundaries

Once a space is created, its boundary is edited differently depending on the current **Auto-Boundary** setting.

- If **Auto-Boundary** is enabled, the space is associated with a set of walls, regardless of whether the walls are visible. When the walls that surround the space change, in many cases, the space updates automatically.
- If **Auto-Boundary** is disabled, the space must be edited directly as a path object.

Each space has a control point that indicates the label location. If the space is auto-bounded, it also has a separate "anchor" control point that locates the space within its boundary. The anchor control point is typically in the center of the space's bounding box, so it may be directly on top of the label control point. (If you create the space with Inner Boundary mode, the anchor is placed where you click to create the space.)

To move a label or an anchor, click the control point and drag it to a new location; click again to move the point. If a label is moved outside of the space boundary, a line is drawn to connect the label to the space automatically.



Click the control point of the space label and drag it outside the space boundary

Click to place the label; a line is automatically drawn from the space boundary to the new position of the control point

To edit a space with **Auto-Boundary** enabled:

1. Add, delete, or move the associated walls.
2. The spaces and labels update automatically, unless you are adding a new wall that affects an existing space or removing a wall that surrounds a space.
   - If walls have been moved, the wall settings (thickness or components) have changed, or you move, edit, or delete an associated column or wall feature, the associated space object(s) and labels update automatically.
   - If walls that are associated with a space are deleted, the space changes from an auto-bound space to a manual space. The space can be associated with new walls.
   - If walls have been added, update the space object(s). Select one or more spaces, and then click **Update Boundary** from the Object Info palette. The location of the updated space is based on the position of the anchor point; other areas created by the dividing wall remain empty.

   Alternatively, select **AEC > Space Planning > Update Space Boundaries** to update all spaces, whether or not they are selected.

3. The boundary of the space adjusts to match the surrounding walls; if the space label anchor point is now outside the space, a line is drawn automatically from the label to the space.

   In cases where architects may have adjacent spaces in a design that are not explicitly separated by an actual wall, the <Virtual wall> wall style (a wall without width or height) gives the user a boundary for space objects to see when Auto-Boundary is enabled, primarily for IFC export.

To edit a space with **Auto-Boundary** disabled:

- To add, subtract, and change vertices, edit the polyline with the **Reshape** tool.
- The space object is a path object and some parameters cannot be edited directly from the Object Info palette. To make changes to the space path object, including changing its dimensions, select **Modify > Edit Space**. For example, you can reshape, add, clip, intersect, and combine into surfaces. Click **Exit Path** to return to the drawing.

Converting Space Boundary Associations

Converting to Manual Mode

Auto-bounded spaces can be converted to manual mode, releasing them from their association with walls.

To convert auto-bounded spaces to manual spaces:

1. Select the auto-bounded space. In the Object Info palette, the **Auto-Boundary** mode displays Auto-Bounded.
2. Select Manual from the **Auto-Boundary** list.

   The space is disassociated from its wall set, though it remains in place.

Converting to Auto-Bounded Mode Using Anchor Points

Manually bounded spaces can be converted to auto-bounded mode by automatically associating them with the surrounding walls.

To convert manual spaces to auto-bounded spaces:

1. Select the manual space(s). In the Object Info palette, the **Auto-Boundary** mode displays Manual.
2. Select Auto-Boundary from the **Auto-Boundary** list.

   The space object(s) is regenerated using the anchor point(s) to find the surrounding walls and associate them with the space(s).

Converting to Auto-Bounded Mode by Picking Boundaries

Manually drawn spaces can be associated with a set of walls, converting them to auto-bounded mode. Similarly, an auto-bounded space can be associated with a new set of walls by this method.

To convert manual spaces to auto-bounded spaces:

1. Select the manual space. In the Object Info palette, the **Auto-Boundary** mode displays Manual.
2. Select Pick Boundary from the **Auto-Boundary** list.

   The **Space** tool is automatically activated, and Inner Boundary mode is selected (Picked Walls mode can also be selected).

3. For Inner Boundary mode, click in an open area of the drawing that is bounded by walls. For Picked Walls mode, click on each wall that forms the wall set and press Enter, or click the check mark button on the Tool bar.
4. The manual space is associated with the wall set. Its location and size may change as it is bounded by the wall set.

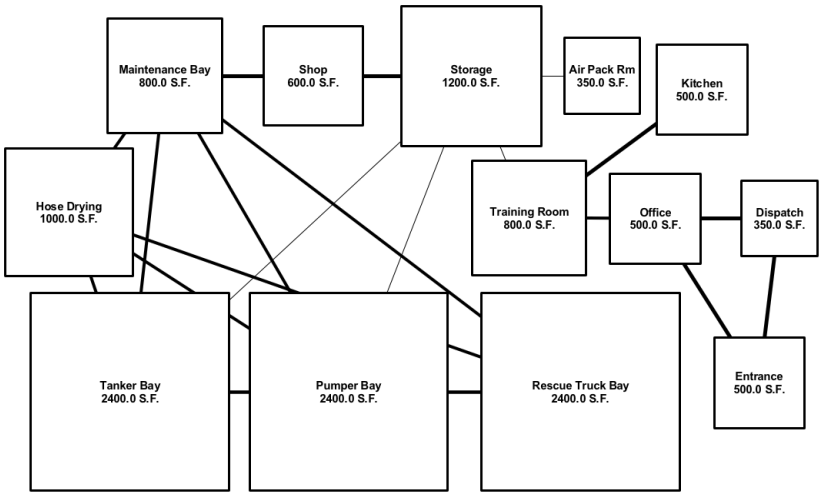
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Programming Studies

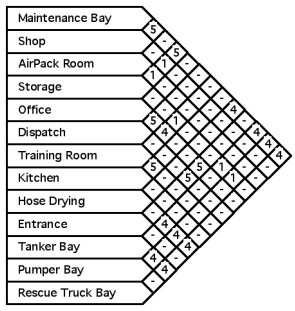
The Vectorworks Architect product provides several tools for creating and managing space programming study information in both tabular and graphical formats, including an adjacency matrix, bubble diagram, and stacking diagram.

In an adjacency matrix worksheet, information is used to automatically create the following items in the drawing:

- a bubble diagram, containing space objects for each space in the matrix, with the appropriate area and department information, and space links connecting the spaces, with line weights indicating the strength of the relationship between the spaces
- an adjacency matrix diagram
- an adjacency score object, showing the total lengths of all the space links
- a stacking diagram



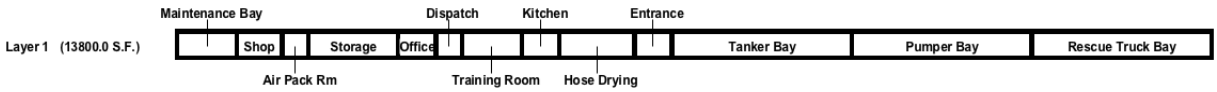
Bubble diagram



Adjacency matrix

Score: 61940

Adjacency score



Stacking diagram

The space objects are square when they are first created, but they can be reshaped with the **Reshape** tool. The spaces can be repositioned to create clusters of related spaces.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating an Adjacency Matrix
Creating a Bubble Diagram
Creating a Stacking Diagram

Creating an Adjacency Matrix

Adjacency matrices specify, in tabular format, the spaces and area requirements that must be incorporated into a design. In addition, the relationships among the various spaces are identified. Spaces which should be located next to each other are strongly related, while spaces which can or should be separated have a weak relationship.

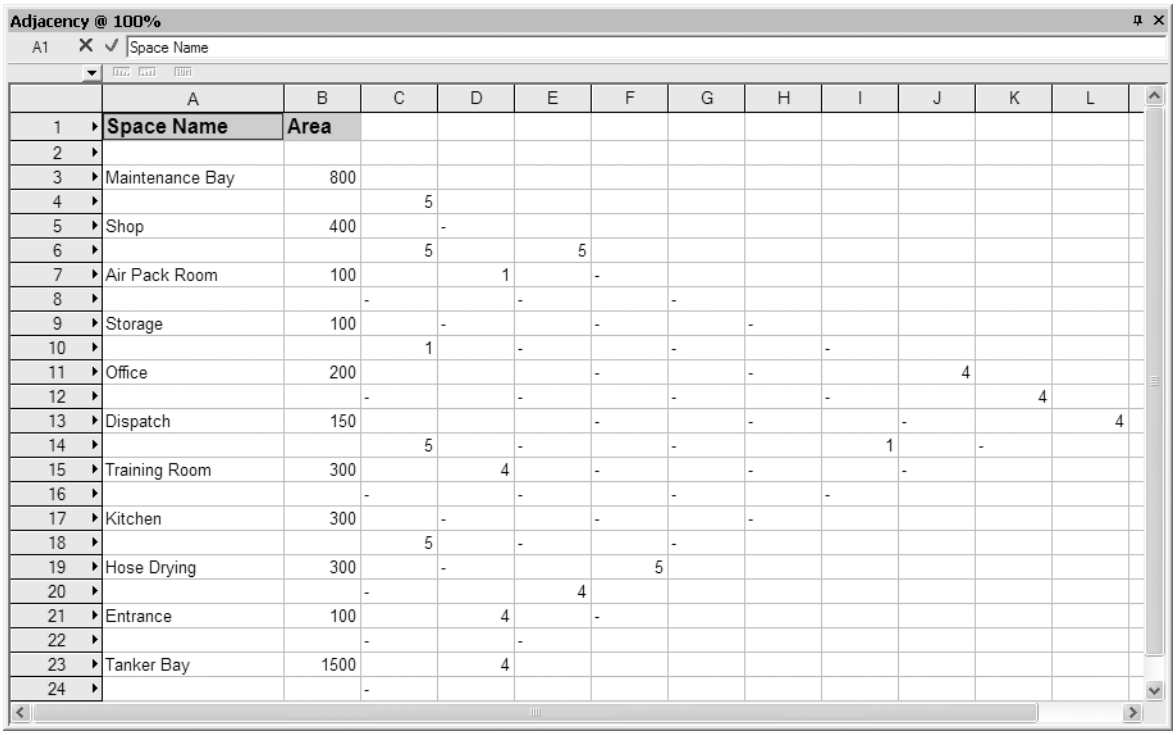
An adjacency matrix is typically developed in a spreadsheet program and imported into a Vectorworks file. An adjacency matrix can also be created in a worksheet in a Vectorworks file, exported as a tab-delimited file, and then imported back into a Vectorworks file to create an adjacency matrix.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating an Adjacency Matrix Worksheet
Importing an Adjacency Matrix Spreadsheet
Inserting an Adjacency Matrix

Creating an Adjacency Matrix Worksheet

To create an adjacency matrix worksheet:

1. From the Resource Browser, select **Resources > New Resource > Worksheet**.
2. Create the adjacency matrix, using every other row for space names and areas, and alternate rows for specifying the relationships among the spaces.



	A	B	C	D	E	F	G	H	I	J	K	L
1	**Space Name**	**Area**										
2												
3	Maintenance Bay	800										
4			5									
5	Shop	400		-								
6			5		5							
7	Air Pack Room	100		1		-						
8			-		-		-					
9	Storage	100		-		-		-				
10			1		-		-		-			
11	Office	200				-		-		4		
12			-		-		-		-		4	
13	Dispatch	150				-		-		-		4
14			5		-		-		1		-	
15	Training Room	300		4		-		-		-		
16			-		-		-		-			
17	Kitchen	300		-		-		-				
18			5		-		-					
19	Hose Drying	300		-			5					
20			-		4							
21	Entrance	100		4		-						
22			-		-							
23	Tanker Bay	1500		4								
24			-									

Any third-party spreadsheet program that can save a tab- or comma-delimited text file can be used to create the adjacency matrix. Ensure that there is at least one cell containing data on each spreadsheet row. Certain spreadsheet programs do not export rows without data, which could disrupt the expected sequence of data in the file as it is imported into the Vectorworks file.

3. Click **File > Export > Export Worksheet** to export the worksheet as a tab- or comma-delimited file.
4. Click **OK** and specify the name and location of the file.

Creating Worksheets
Importing an Adjacency Matrix Spreadsheet

Importing an Adjacency Matrix Spreadsheet

To import an adjacency matrix:

1. Ensure that the file to be imported is a tab- or comma-delimited file.

2. Select **AEC > Space Planning > Import Adjacency Matrix**. The Import Adjacency Matrix dialog box opens. Specify the adjacency matrix file import options.

Click to show/hide the parameters.

Parameter	Description
External Filename	Enter the name of the adjacency matrix file or click **Browse** to specify its location; a sample file is specified by default
Header Rows	Specifies the number of rows in the spreadsheet that do not include space or space relationship information
Row Spacing	Set this to 0 (zero) if there is a space name on every row or set this to 1 if there is a space name on every other row in the drawing file
Field Delimiter	Specifies the delimiter that was used to separate fields for the spreadsheet
Column Numbers and Preview	Specifies the columns to include in the adjacency matrix as well as their position
Department	Select to include department number information and specify its column number
Space	Select to include area name information and specify its column number
Area	Select to include the space area information and specify its column number
Relationships	Select to include the space relationship information and specify its column number
Preview	Displays the data in the selected columns

3. Click **OK** to import the adjacency matrix. If this is the first time an adjacency matrix, stacking diagram, spaces, or space links are placed in the file, the object properties dialog box opens for each type of object. Accept the default values and click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting an Adjacency Matrix

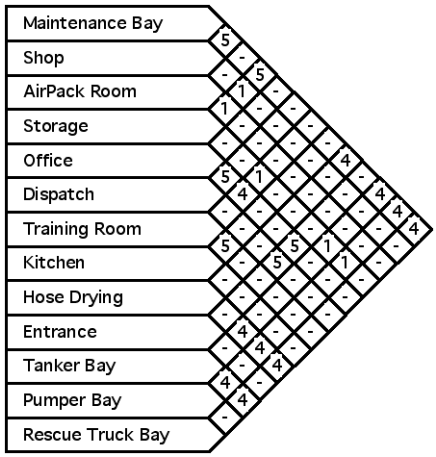
Inserting an Adjacency Matrix

An adjacency matrix diagram can also be created manually with the **Adjacency Matrix** tool, based on spaces and space relationships currently in the file.

To insert an adjacency matrix diagram:

1. Click the **Adjacency Matrix** tool from the Space Planning tool set.
2. Click at the desired location for the adjacency matrix.
3. Click again to set the matrix rotation. If this is the first time an adjacency matrix diagram is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all adjacency matrix diagrams placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

   The adjacency matrix is created. If no space objects currently exist in the drawing, an adjacency matrix place holder is created.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



The adjacency matrix is more than just a presentation graphic, because it dynamically shows what is in the drawing. The adjacency matrix updates automatically when spaces or space links change.

The **Rotation**, **Height Scale** and **Width Scale** parameters of an adjacency matrix can be edited in the Object Info palette.

To adjust the order of adjacency matrix spaces, change the **Matrix Order** number in the Object Info palette for the relevant space object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Space Properties
Creating an Adjacency Matrix

Creating a Bubble Diagram

A bubble diagram is a graphical view of the data from a space planning spreadsheet. Spaces are connected by space links that represent the relationships among the spaces. Thick links indicate a strong relationship between the spaces, while weak relationships are drawn with thin lines. The efficiency of the bubble diagram layout can be determined by calculating its adjacency score.

A bubble diagram is inserted into a drawing automatically when an adjacency matrix is imported (see “Importing an Adjacency Matrix Spreadsheet” on page 438). A bubble diagram can also be created manually by linking spaces with the **Space Link** tool.

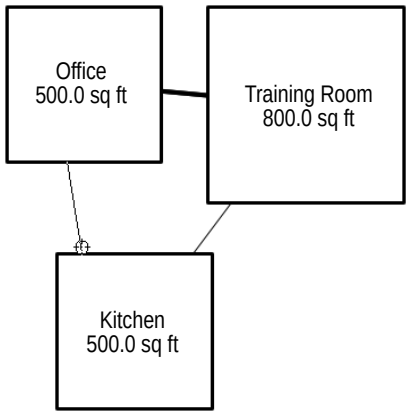
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Linking Spaces
Calculating the Adjacency Score

Linking Spaces

To link spaces:

1. Create spaces as described in “Creating Spaces with the Space Tool” on page 403.
2. Click the **Space Link** tool from the Space Planning tool set.
3. Click on the first space to link, and then on the second space. The space link is automatically created.



The space links update dynamically as the spaces are moved around in the drawing.

To change the strength of a link, select it and change the **Strength** value (from 1 to 18) in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Calculating the Adjacency Score

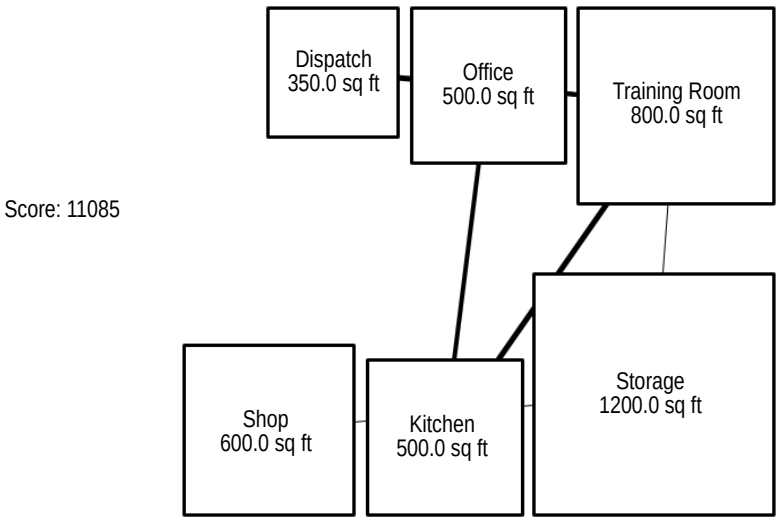
Calculating the Adjacency Score

The adjacency score object displays a numerical value that represents the efficiency of the space layout. The score value is determined by first multiplying the link length times strength for each space link, and then calculating the sum of these values for all the space links in the file. The lower the score, the more efficient the layout.

To insert an adjacency score object:

1. Click the **Adjacency Score** tool from the Space Planning tool set.
2. Click in the document to set the object location.
3. Click a second time to set the object rotation.

   If spaces or space link objects are deleted, the adjacency score object value is not automatically updated. Slightly move any space or adjacency score object in the document to reset the adjacency score object value.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Bubble Diagram

Creating a Stacking Diagram

A stacking diagram provides a floor-by-floor representation of the total amount of area taken up by all the spaces on the floors. It is useful for balancing the allocation of spaces among several floors.

A stacking diagram is inserted into a drawing automatically when an adjacency matrix is imported (see “Importing an Adjacency Matrix Spreadsheet” on page 438). A stacking diagram can also be inserted manually with the **Stacking Diagram** tool. The stacking diagram is based on spaces and space relationships currently in the file.

To insert a stacking diagram:

1. Click the **Stacking Diagram** tool from the Space Planning tool set.
2. Click at the desired location for the stacking diagram. Click again to set the diagram rotation. If this is the first time a stacking diagram is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all stacking diagrams placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.
3. The stacking diagram is created.

   The layers are “stacked” on the basis of their layer elevation values, which can be changed in the Organization dialog box (**Tools > Organization**).

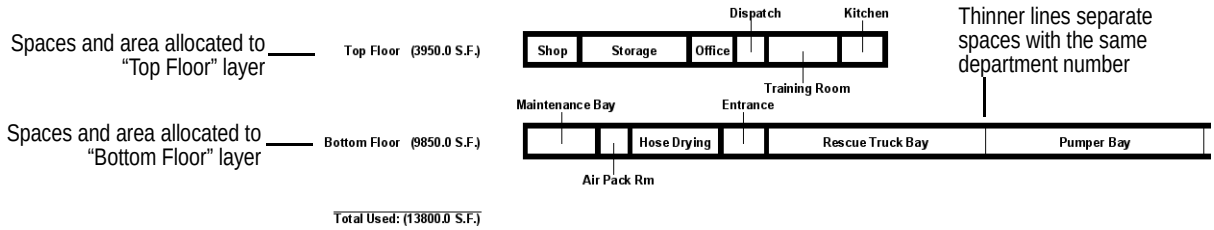
   If no space objects currently exist in the drawing, a stacking diagram place holder is created.

Like the adjacency matrix, the stacking diagram dynamically shows what is in the drawing. The stacking diagram updates automatically when spaces or space links are changed. In addition, fill attributes assigned to spaces are reflected in the stacking diagram automatically.

The **Rotation**, **Height Scale** and **Width Scale** parameters of an adjacency matrix can be edited in the Object Info palette. In addition, the **Gap Scale** parameter controls the distance between the stacking diagram rows. **Show Totals** toggles the display of the total amount of space used.

The stacking diagram is arranged in rows that correspond to the layers of the drawing. The spaces present in each layer are shown in separate rows; the layer name and total area used are displayed to the left of each row.

Spaces in the stacking diagram are grouped by **Department** name (set for each space in the Object Info palette). Adjacent spaces with the same **Department** name are separated by thin lines.



Structural Elements

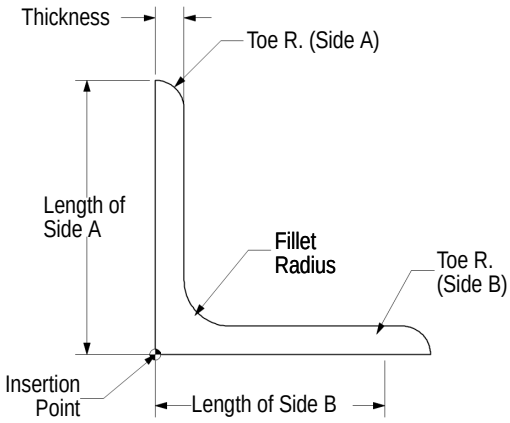
Structural Shapes and Details

Several commonly used structural steel shapes are available as 2D objects. 3D equivalents of these objects are also available in the Vectorworks Design Series products. Since the 2D and 3D parameters are nearly identical, all parameters are documented with the 2D objects for convenience. Two additional 2D/3D structural shapes are also available in the Vectorworks Design Series products: bulb flat and Z-section, as well as four additional series (ANZ, BSI, DIN, and JIS).

> The **Update Plug-in Objects** command may need to be run on files containing structural shapes that were created in an earlier version of Vectorworks software. This command converts the structural shapes to the latest format; see “Migrating from Previous Versions” on page 28.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Angle



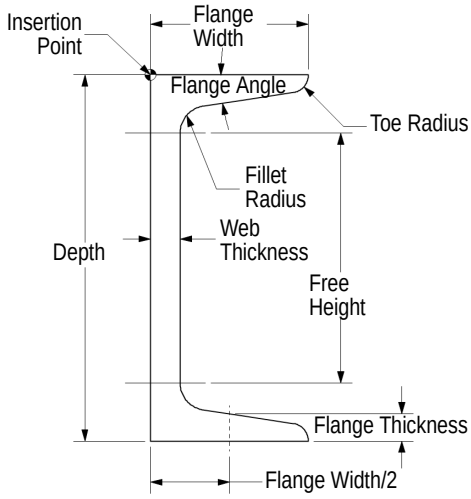
To create an angle object:

1. Click the **Angle** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object’s rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the angle properties and click **OK**.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the angle size
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D angle
Length (Vectorworks Design Series required)	Enter the length of the 3D angle
Custom Size	Select this option to enable fields for specifying a custom angle size
Length of Side A/B	Displays the length of angle sides A and B, or if **Custom Size** is selected, allows entry of a custom value
Thickness	Displays the thickness of the angle sides, or if **Custom Size** is selected, allows entry of a custom value
Toe R. (Side A/B)	Displays the toe radius value for sides A and B, or if **Custom Size** is selected, allows entry of a custom value
Fillet Radius	Displays the fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the angle area
About Axis X-X/Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
About Axis Z-Z	
Radius of Gyration	Displays the least radius of gyration about the principal (Z-Z) axis
Tan (Alpha)	Displays the tangent of the angle of the principal axis from the vertical

Channel



To create a channel object:

1. Click the **Channel** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the channel properties and click **OK**.

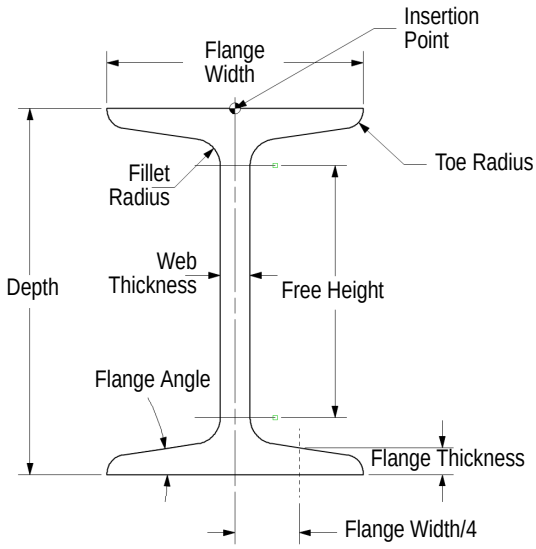
Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the channel size
Draw Free Height Lines	Select whether to draw the 2D channel with free height lines
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D channel
Length (Vectorworks Design Series required)	Enter the length of the 3D channel
Custom Size	Select this option to enable fields for specifying a custom channel size
Depth	Displays the depth between endpoints, or if **Custom Size** is selected, allows entry of a custom value
Flange Width	Displays the flange width, or if **Custom Size** is selected, allows entry of a custom value
Flange Thickness	Displays the flange thickness, or if **Custom Size** is selected, allows entry of a custom value
Flange Angle (Deg.)	Displays the flange angle degrees, or if **Custom Size** is selected, allows entry of a custom value
Web Thickness	Displays the web thickness of the channel, or if **Custom Size** is selected, allows entry of a custom value

Parameter	Description
Toe Radius	Displays the toe radius value, or if **Custom Size** is selected, allows entry of a custom value
Fillet Radius	Displays the fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the channel area
About Axis X-X/Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

Structural Shapes and Details

I-beam



To create an I-beam object:

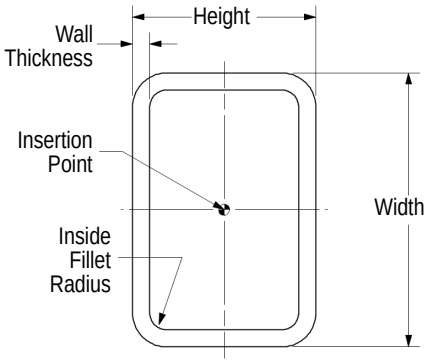
1. Click the **I-Beam** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the I-beam properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the I-beam size
Length (Vectorworks Design Series required)	Enter the length of the 3D I-beam
Draw Center Line	Select whether to draw the 2D I-beam with center lines
Draw Free Height Lines	Select whether to draw the 2D I-beam with free height lines
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D I-beam
Custom Size	Select this option to enable fields for specifying a custom I-beam size
Depth	Displays the depth between endpoints, or if **Custom Size** is selected, allows entry of a custom value
Flange Width	Displays the flange width, or if **Custom Size** is selected, allows entry of a custom value
Web Thickness	Displays the web thickness of the flange, or if **Custom Size** is selected, allows entry of a custom value
Flange Thickness	Displays the flange thickness, or if **Custom Size** is selected, allows entry of a custom value
Flange Angle (Deg.)	Displays the flange angle in degrees, or if **Custom Size** is selected, allows entry of a custom value
Fillet Radius	Displays the fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Toe Radius	Displays the toe radius value, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the I-beam area
About Axis X-X/Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

Structural Shapes and Details

Rectangular Tubing



To create a rectangular tubing object:

1. Click the **Rectangular Tubing** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the rectangular tubing properties and click **OK**.

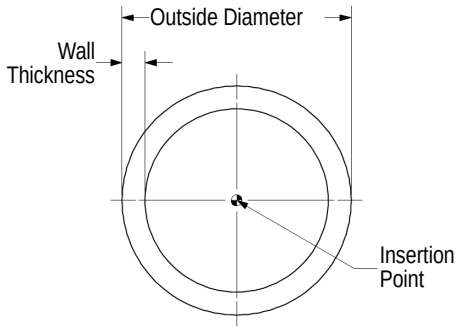
Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the rectangular tubing size
Length (Vectorworks Design Series required)	Enter the length of the 3D rectangular tubing
Draw Center Lines	Select whether to draw the 2D rectangular tubing with center lines
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D rectangular tubing
Custom Size	Select this option to enable fields for specifying a custom rectangular tubing size
Width	Displays the width of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Height	Displays the height of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Wall Thickness	Displays the wall thickness of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Inside Fillet Radius	Displays the inside fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the rectangular tubing area

Parameter	Description
About Axis X-X/ Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

Structural Shapes and Details

Round Tubing



To create a round tubing object:

1. Click the **Round Tubing** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the round tubing properties and click **OK**.

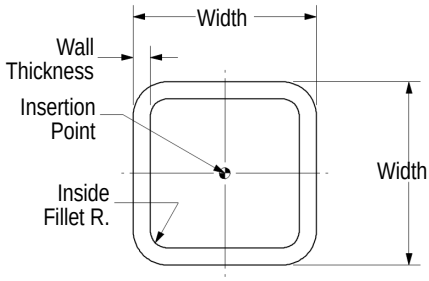
Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the round tubing size
Length (Vectorworks Design Series required)	Enter the length of the 3D round tubing
Draw Center Lines	Select whether to draw the 2D round tubing with center lines
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D round tubing
Custom Size	Select this option to enable fields for specifying a custom round tubing size

Parameter	Description
Outside Diameter	Displays the outside diameter of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Wall Thickness	Displays the wall thickness of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the round tubing area
About Axis X-X/Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

Structural Shapes and Details

Square Tubing



To create a square tubing object:

1. Click the **Square Tubing** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the square tubing properties and click **OK**.

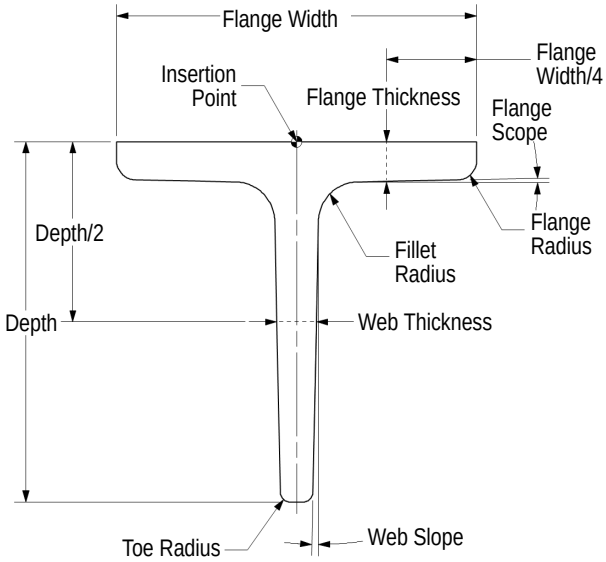
Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the square tubing size

Parameter	Description
Length (Vectorworks Design Series required)	Enter the length of the 3D square tubing
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D square tubing
Draw Center Lines	Select whether to draw the 2D square tubing with center lines
Custom Size	Select this option to enable fields for specifying a custom square tubing size
Width	Displays the width of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Wall Thickness	Displays the wall thickness of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Inside Fillet R.	Displays the inside fillet radius value of the tubing, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the square tubing area
About Axis X-X/Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

Structural Shapes and Details

Tee



To create a tee object:

1. Click the **Tee** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the tee properties and click **OK**.

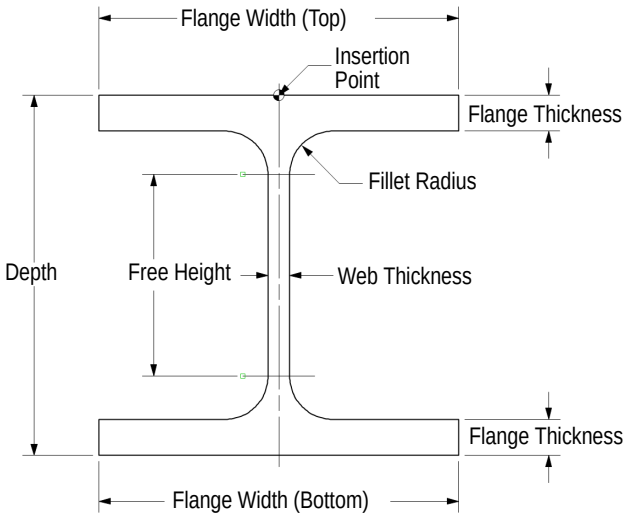
Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the tee size
Length (Vectorworks Design Series required)	Enter the length of the 3D tee
Draw Center Line	Select whether to draw the 2D tee with center lines
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D tee
Custom Size	Select this option to enable fields for specifying a custom tee size
Depth	Displays the depth between endpoints, or if **Custom Size** is selected, allows entry of a custom value
Flange Width	Displays the flange width, or if **Custom Size** is selected, allows entry of a custom value
Web Thickness	Displays the web thickness, or if **Custom Size** is selected, allows entry of a custom value
Flange Thickness	Displays the flange thickness, or if **Custom Size** is selected, allows entry of a custom value
Web Slope (Deg.)	Displays the web slope in degrees, or if **Custom Size** is selected, allows entry of a custom value

Parameter	Description
Flange Slope (deg.)	Displays the flange slope in degrees, or if **Custom Size** is selected, allows entry of a custom value
Toe Radius	Displays the toe radius value, or if **Custom Size** is selected, allows entry of a custom value
Flange Radius	Displays the flange radius value, or if **Custom Size** is selected, allows entry of a custom value
Fillet Radius	Displays the fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the tee area
About Axis X-X/Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

Structural Shapes and Details

Wide Flange



To create a wide flange object:

1. Click the **Wide Flange** tool from the Detailing tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the wide flange properties and click **OK**.

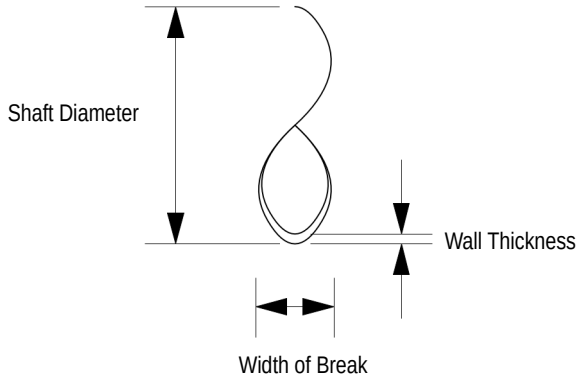
Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the wide flange size
Length (Vectorworks Design Series required)	Enter the length of the 3D wide flange
Draw Center Line	Select whether to draw the 2D wide flange with center lines
Draw Free Height Lines	Select whether to draw the 2D wide flange with free height lines
Place Locus at Centroid	Select whether to draw a locus at the centroid of the 2D wide flange
Custom Size	Select this option to enable fields for specifying a custom wide flange size
Depth	Displays the depth between endpoints, or if **Custom Size** is selected, allows entry of a custom value
Flange Width (Top) / (Bottom)	Displays the top and bottom flange width of the wide flange, or if **Custom Size** is selected, allows entry of a custom value
Web Thickness	Displays the web thickness, or if **Custom Size** is selected, allows entry of a custom value
Flange Thickness	Displays the flange thickness, or if **Custom Size** is selected, allows entry of a custom value
Fillet Radius	Displays the fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the wide flange area
About Axis X-X/Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

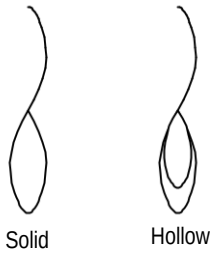
Shaft Break

To insert a Shaft Break object:

1. Click the **Shaft Break** tool from the Detailing tool set.
2. Click to define the start point, click again to define the end point and rotation, and drag the cursor and click to set the width.



Click to show/hide the parameters.

Parameter	Description
Shaft Diameter	Specify the diameter of the shaft break object
Width of Break	Specify the width of the shaft break object
Configuration	Select **Solid** or **Hollow** 
Wall Thickness	For hollow shaft break configurations, indicate the thickness of the shaft wall

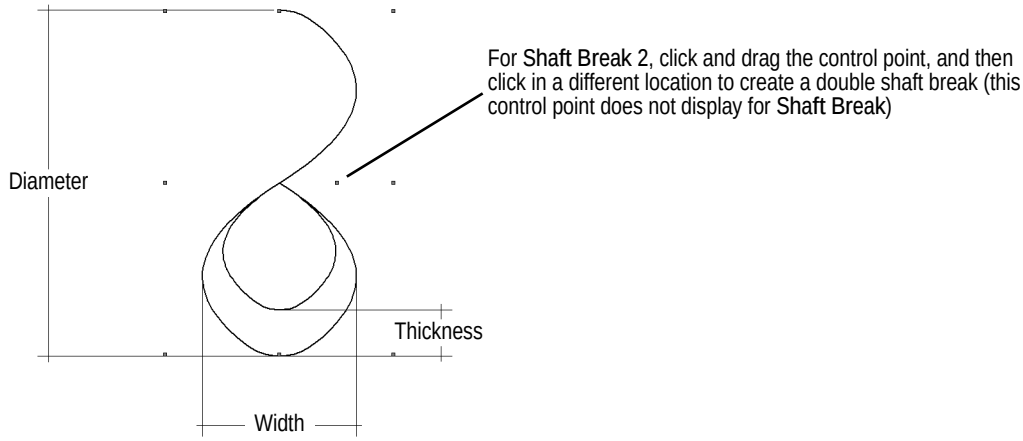
Structural Shapes and Details

Shaft Break 2

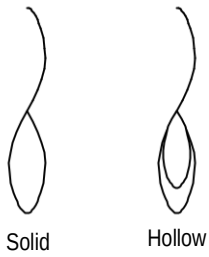
To insert a shaft break 2 object.

1. Click the **Shaft Break 2** tool from the Detailing tool set.
2. Click to define the start point, click again to define the end point and rotation, and drag the cursor and click to set the width. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.

3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Shaft Diameter	Specify the diameter of the shaft break object
Width of Break	Specify the width of the shaft break object
Configuration	Select **Solid** or **Hollow** 
Wall Thickness	For hollow shaft break configurations, indicate the thickness of the shaft wall

Structural Shapes and Details

Slot

To insert a slot:

1. Click the **Slot** tool from the Detailing tool set.
2. Click to define the start point of the slot, click again to define the end point and rotation, and drag the cursor and click to set the slot width.

Click to show/hide the parameters.

Parameter	Description
Width	Specify the width of the slot object
Length	Specify the width of the slot object
Show Center Line	When selected, draws the slot with center line(s)

Structural Shapes and Details

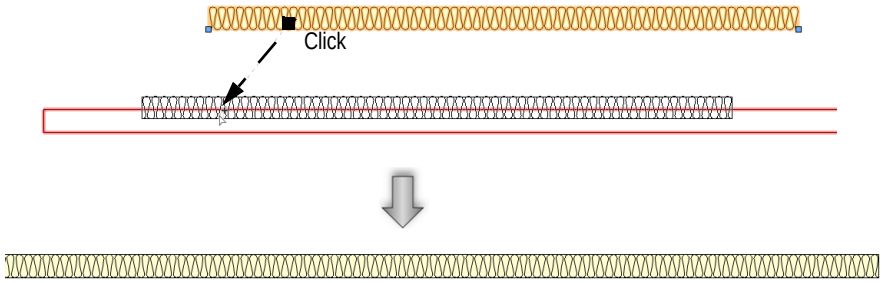
Batt Insulation

The **Batt Insulation** tool draws a planar insulation fill along a line. This tool is useful for creating custom details. The insulation can be placed within a wall, where it automatically adjusts to the length of the wall.

A better method of creating insulation fills, especially for walls, is to use a tile fill, which scales and rotates within walls and slab and wall components.

 To create batt insulation:

1. Click the **Batt Insulation** tool from the Detailing tool set.
2. Click to mark the beginning of the insulation fill and then click again to mark the end of the object. If this is the first time the tool is used in this session, the Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**. The parameters can be changed later in the Object Info palette.
3. The created batt insulation can be dragged into an existing wall.



Click Set Position for a Batt Insulation in Wall object, to fit the insulation to the wall; Size to Wall Length must be enabled

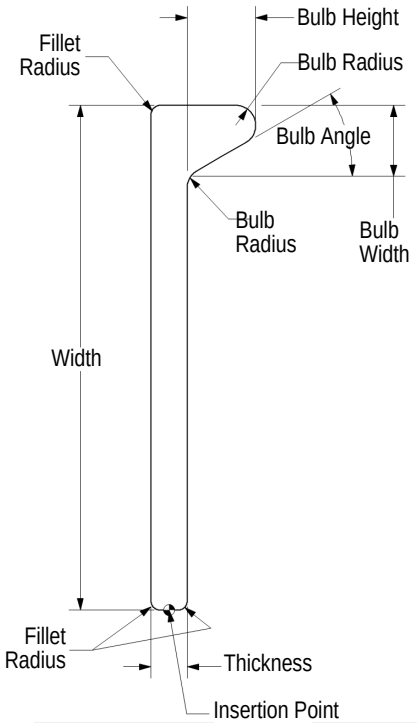
Click to show/hide the parameters.

Parameter	Description
Insert	When inserted in walls, aligns the insertion point of the batt insulation with the containing wall's center line or left or right edge
Break	When inserted in walls, select the type of wall break for the batt insulation
Height	When inserted in walls, sets the height of the insulation from the bottom of the wall
Flip	When inserted in walls, flips the orientation of the batt insulation
Set Position	When inserted in walls and with **Size to Wall Length** selected, sets the insulation to fit along the length of the wall. The **Move By Points** tool is activated automatically; if **Size to Wall Length** is deselected, set the offset of the batt insulation along the wall as described in "Moving Symbols in Walls with the Reference Point Mode" on page 259.

Parameter	Description
Length	Specifies the length of the batt insulation, when **Size to Wall Length** is deselected
Thickness	Sets the width of the batt insulation
Offset	Sets the offset of the batt insulation from its insertion point
Folds/Unit Thk.	Sets the thickness of the insulation folds
Size to Wall Length	When inserted in walls, automatically sets the length of the batt insulation to match the wall length
Add to Start/End	Specifies an offset from the start or end of the wall, when the batt insulation is sized to fit the wall length
Solid Fill	Applies a solid fill to the background of the batt insulation; set the fill color from the Attributes palette

Defining Tiles
Structural Shapes and Details

Bulb Flat



To create a bulb flat object:

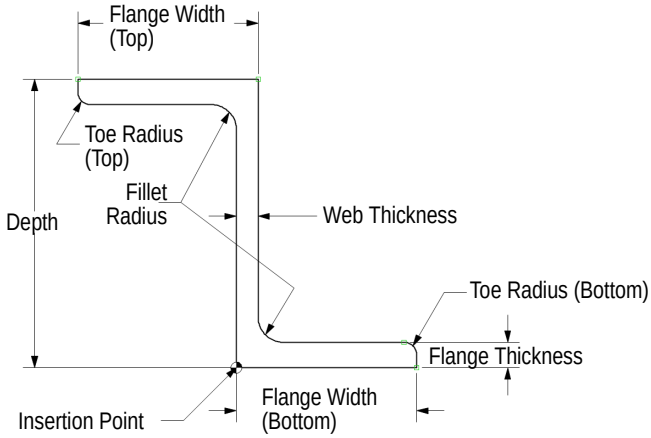
1. Click the **Bulb Flat** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the bulb flat properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the bulb flat size
Length (3D only)	Enter the length of the 3D bulb flat
Place Locus at Centroid (2D only)	Select whether to draw a locus at the centroid of the 2D bulb flat
Custom Size	Select this option to enable fields for specifying a custom bulb flat size
Width	Displays the bulb flat width, or if **Custom Size** is selected, allows entry of a custom value
Thickness	Displays the bulb flat thickness, or if **Custom Size** is selected, allows entry of a custom value
Bulb Height	Displays the bulb height, or if **Custom Size** is selected, allows entry of a custom value
Bulb Width	Displays the bulb width, or if **Custom Size** is selected, allows entry of a custom value
Bulb Radius	Displays the bulb radius value, or if **Custom Size** is selected, allows entry of a custom value
Fillet Radius	Displays the fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Bulb Angle (Deg.)	Displays the bulb angle in degrees, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter units for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the bulb flat area
About Axis X-X/ Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

Structural Shapes and Details

D Z-Section



To create a Z-section object:

1. Click the **Z-Section** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the Z-section properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Series	Select the desired series to display the appropriate nominal sizes
Size	Select the Z-section size
Length (3D only)	Enter the length of the 3D Z-section
Place Locus at Centroid (2D only)	Select whether to draw a locus at the centroid of the 2D Z-section
Custom Size	Select this option to enable fields for specifying a custom Z-section size
Depth	Displays the depth between endpoints, or if **Custom Size** is selected, allows entry of a custom value
Flange Width (Top) / (Bottom)	Displays the top and bottom flange width, or if **Custom Size** is selected, allows entry of a custom value
Web Thickness	Displays the web thickness, or if **Custom Size** is selected, allows entry of a custom value
Flange Thickness	Displays the flange thickness, or if **Custom Size** is selected, allows entry of a custom value
Toe Radius (Top) / (Bottom)	Displays the top and bottom toe radius value, or if **Custom Size** is selected, allows entry of a custom value

Parameter	Description
Fillet Radius	Displays the fillet radius value, or if **Custom Size** is selected, allows entry of a custom value
Section Properties	Section properties are automatically calculated and displayed based on inch units for imperial shapes and millimeter for metric shapes (regardless of the units selected and **Show Unit Mark** setting in the Units dialog box)
Area	Displays the Z-section area
About Axis X-X/ Y-Y	
Moment of Inertia	Displays the moment of inertia about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Section Modulus	Displays the section modulus about both the horizontal (X-X) and vertical (Y-Y) centroidal axes
Radius of Gyration	Displays the radius of gyration about both the horizontal (X-X) and vertical (Y-Y) centroidal axes

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Structural Shapes and Details

Framing

The Vectorworks Architect software includes a suite of tools for developing framing plans. There are tools for framing roofs, walls, and floors. Both architects and builders can take advantage of these tools to visualize framing details while still in the design stage of a project, to generate framing detail drawings, and to develop material take-off lists. The **Simple Beam Calculator** command is also available for analysis of a simply-supported beam with a single load.

A roof face area worksheet can also be added to the drawing from the **VA Create Schedule** command or the Resource Browser. From the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder that is included with the Vectorworks Architect product (see “Resource Libraries” on page 219). Drag the Roof Face Area worksheet to the drawing. The worksheet is populated with information from the objects in the current drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Framing a Roof
Creating Framing Members
Framing a Floor
Framing a Wall
Simple Beam Calculator

Framing a Roof

A roof object can be automatically framed with rafters, collars (collar beams), ridges, and other roof elements with the **Roof Framer** command. The command creates framing member objects. If **Enable Auto-classing** is selected in the Standard Naming dialog box, all roof framing members will be automatically placed in the Structural-Framing class.

This command does not work on roof face objects.

To automatically frame a roof:

1. Select the roof object.
2. Select **AEC > Framing > Roof Framer**.

The Roof Framer dialog box opens.

3. Select the tab for the type of roof framing element to create, and select the check box to create the framing element. More than one type can be selected at the same time; other framing elements can be added later.

   An individual framing element can be added with the **Framing Member** tool, available from the Detailing tool set.

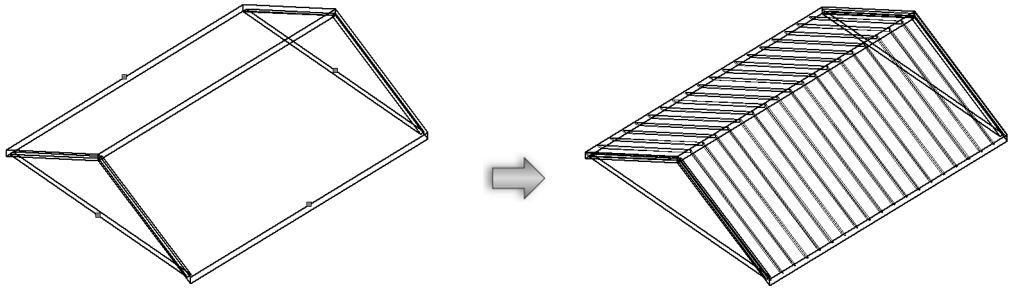
4. Enter the framing parameters for each desired framing element as described in the following sections.
5. For each selected roof framing element, click **Object Attributes** to specify its appearance.

   The Attributes dialog box opens. See “Applying Object Attributes” on page 1093 for information on applying graphic attributes, and “Applying and Mapping Textures” on page 1527 for information on texture parameters.

Click to show/hide the parameters.

Parameter	Description
Graphic Attributes	Each graphic attribute can be set by the class style or to specific settings. If set by the class style, the attributes are controlled by the class selected in the Roof Framer dialog box for the framing element.
Fill	Select Class Style to use the class style for the fill attribute, choose None to apply no fill, or choose a fill style
Pen	Select Class Style to use the class style for the pen attribute, choose None to apply no pen, or choose a solid/dash or pattern style
Line Thickness	Select Class Thickness to use the class style for the line thickness attribute, choose Set Thickness to specify the line thickness, or select a line thickness
Texture (Renderworks required)	Select Class to use the class texture for the framing element (see “Applying Object Textures by Class” on page 1542), choose None to apply no texture, or choose Select to set the texture parameters
Texture	Select the texture to apply from either the default content or the current file’s content; see “Resource Libraries” on page 219
Map Type	Select the texture map type
Radius	For sphere and cylinder maps, sets the texture radius; the default radius is the same as the 3D object radius. Increasing this value reduces the size of the texture on the object.
Start/End Cap	Applies the texture to the start and end caps of the framing element
Repeat Horizontally/ Vertically	Repeats the texture in a horizontal and/or vertical direction
Scale Factor	Enter a scale value to determine the texture size when projected onto the object
Horizontal/Vertical Offset	Sets the start location of the texture horizontally and vertically
Rotation	Sets the angle of texture rotation; enter a rotation value from 0 to 360 degrees

6. Click **OK** to return to the Roof Framer dialog box.
7. Click **OK** to frame the roof.



The Attributes Palette
Creating Framing Members
Creating Rafters
Creating Collars
Creating Plates
Creating Purlins
Creating Ridges
Creating Trimmers
Creating Hip and Valley Rafters
Custom Frame Element Sizes

A Creating Rafters

To create rafters:

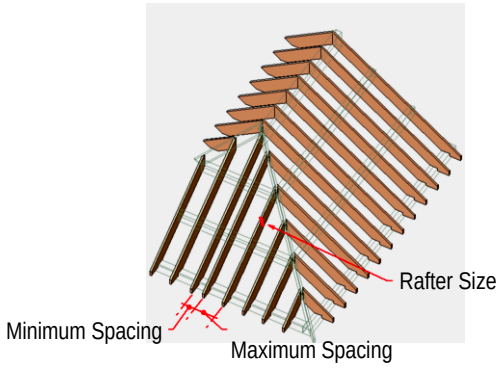
1. In the Roof Framer dialog box, click the Rafters tab, and select **Create Rafters**.
2. Specify the rafter parameters.

Click to show/hide the parameters.

Parameter	Description
Rafter Size	Specifies the rafter size in width x height, in current file units. Sizes can be edited by selecting **Edit List** (see "Custom Frame Element Sizes" on page 470).
Dormer Rafter Size	If the roof contains dormer windows, enter the size, in Width x Height, of the dormer rafters
Minimum Spacing	Specifies the minimum distance allowed between rafters
Maximum Spacing	Specifies the maximum distance allowed between rafters; for consistent rafter spacing of the entire roof, the **Minimum** and **Maximum Spacing** values should be equal
Set top of Rafters to	
Top of Roof object	Matches the tops of the rafters to the top of the roof object
Bottom of Roof object	Treats the roof object as a roof deck and matches the tops of the rafters to the bottom of the roof
Match Rafters across Ridges/Hips/Valleys	Select to ensure that opposing rafters meet on the roof ridges, hips, and valleys

Parameter	Description
Class	Select a class for the roof element to control its appearance and visibility. The classes present in the drawing are listed; alternatively select the currently active class or create a new class from the Organization dialog box by clicking **Classes** (see "Classes" on page 176).
Classes	Edits class attributes
Object Attributes	Specifies the rafter attributes

3. Click **OK** to create the rafters.



Framing a Roof
Setting Class Properties
Custom Frame Element Sizes

Creating Collars

To create collars (collar beams):

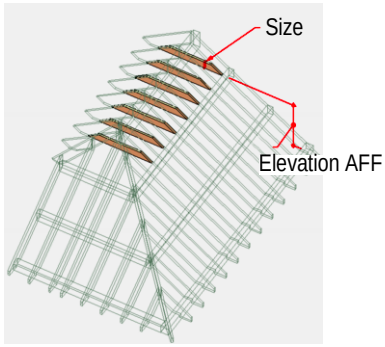
1. In the Roof Framer dialog box, click the Collars tab, and select **Create Collars**.
2. Specify the collar parameters.

Click to show/hide the parameters.

Parameter	Description
Style	Select the collar style; flush collars or collar ties will only connect to the common rafters, joist collars connect to both common and jack rafters • Joist: tie beam that rests on the bearing wall and overlaps the rafters • Flush Collar: tie collar that is flush to the rafters, usually attached to the rafters with metal plates • Collar Tie: tie collar that scabs onto the rafters and is attached with nails, bolts, and so on
Size	Specifies the collar size in width x height, in current file units. Sizes can be edited by selecting **Edit List** (see "Custom Frame Element Sizes" on page 470).
Spacing	Select the collar spacing (Every Rafter, Every Other Rafter, and Every Third Rafter)

Parameter	Description
Location	For joist or collar tie styles, specifies the location of the collars in reference to the rafters. For flush collar style, the location is automatically set to Middle. • Middle places one collar at the center of the rafter for flush collar and joist styles, and one collar on each side of the rafter for the collar tie style • Left/Right places a single collar on the designated side of the rafter for joist or collar tie styles. Joists and collar ties extend to the intersection of the rafter tops.
Elevation AFF	Specifies the elevation above finished floor of the bottom of the collars. If Flush Collar or Collar Tie is selected for **Style**, the default elevation is two-thirds of the roof height. Joist style is automatically set to 0.
Class	Select a class for the roof element to control its appearance and visibility. The classes present in the drawing are listed; alternatively select the currently active class or create a new class from the Organization dialog box by clicking **Classes** (see "Classes" on page 176).
Classes	Edits class attributes
Object Attributes	Specifies the collar attributes

3. Click **OK** to create the collars.



Setting Class Properties
Framing a Roof
Custom Frame Element Sizes

Creating Plates

To create plates:

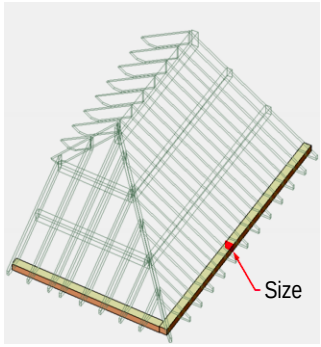
1. In the Roof Framer dialog box, click the Plates tab, and select **Create Plates**.
2. Specify the plate parameters.

Click to show/hide the parameters.

Parameter	Description
Size	Specifies the plate size in width x height, in current file units. Sizes can be edited by selecting **Edit List** (see "Custom Frame Element Sizes" on page 470).
Rafter Connection	
Flush to Top of Plate	Select to set rafters flush to the top of the plate

Parameter	Description
Notched for Plate	Select to notch the rafters to set on the plate; enter the **Notch Depth**, by default this is set to 1.5” The depth of the notch should not exceed one-third of the rafter depth
Class	Select a class for the roof element to control its appearance and visibility. The classes present in the drawing are listed; alternatively select the currently active class or create a new class from the Organization dialog box by clicking **Classes** (see “Classes” on page 176).
Classes	Edits class attributes
Object Attributes	Specifies the plate attributes

3. Click **OK** to create the plates.



Setting Class Properties
Framing a Roof
Custom Frame Element Sizes

Creating Purlins

To create purlins:

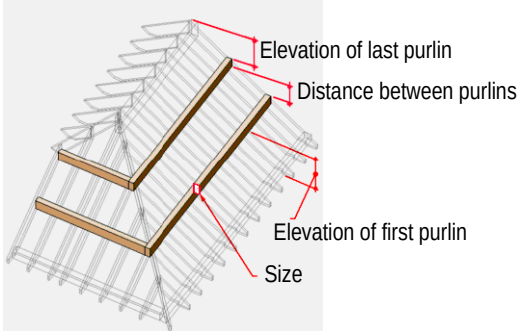
1. In the Roof Framer dialog box, click the Purlins tab, and select **Create Purlins**.
2. Specify the purlin parameters.

Click to show/hide the parameters.

Parameter	Description
Purlin Size	Specifies the purlin size in width x height, in current file units. Sizes can be edited by selecting **Edit List** (see “Custom Frame Element Sizes” on page 470).
Elevation of First Purlin	Specifies the elevation above finished floor of the bottom of the lowest purlin
Elevation of Last Purlin	Sets the elevation from the top of the roof to the last purlin
Create Purlins by:	
Number of Purlins	Indicates the number of purlins to create; the purlins are evenly spaced between the first and last purlins

Parameter	Description
Distance Between Purlins	Specifies the distance between purlins; the number of purlins created is based on this value
Purlin Position	
Top of Rafter	Matches the tops of the purlins to the top of the roof object, as for roof tile supports
Bottom of Rafter	Matches the tops of the purlins to the bottom of the roof, as when the purlins support the rafters
Purlins Start at the Lower End of the Rafter	Sets the elevation of the first purlin to start from the lower end of the rafter
Rotate Purlins to Roof Slope	Rotates the purlins so that they match the slope of the roof
Bearing inset	Specifies the depth at which the purlin is notched into the rafters; if purlins are rotated this field appears dimmed since the inset only applies to horizontal purlins
Class	Select a class for the roof element to control its appearance and visibility. The classes present in the drawing are listed; alternatively select the currently active class or create a new class from the Organization dialog box by clicking **Classes** (see “Classes” on page 176).
Classes	Edits class attributes
Object Attributes	Specifies the purlin attributes

3. Click **OK** to create the purlins.



Setting Class Properties
Framing a Roof
Custom Frame Element Sizes

Creating Ridges

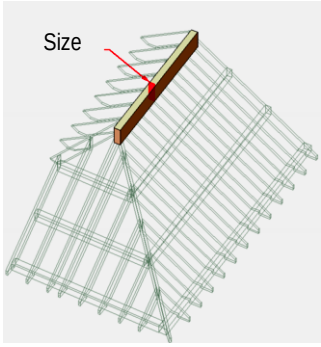
To create ridges:

1. In the Roof Framer dialog box, click the Ridges tab, and select **Create Ridges**.
2. Specify the ridge parameters.

Click to show/hide the parameters.

Parameter	Description
Ridge Size	Specifies the ridge size in width x height, in current file units. Sizes can be edited by selecting **Edit List** (see "Custom Frame Element Sizes" on page 470).
Ridge Elevation	Specifies the ridge elevation options
Flush to Top of Rafters	Indicates that the top of the ridge should be the same as the top of the rafters
Notched into rafters, depth	Specify the depth that the ridge is notched into the rafters
Flush to Bottom of Rafters	Indicates that the bottom of the ridge should be the same height as the bottom of the plumb cut on the rafters
Specify AFF	Indicates the height of the bottom of the ridge, above the finished floor
Flush to Top of Hips	Sets the top of the ridge to be the same as the top of the hips
Class	Select a class for the roof element to control its appearance and visibility. The classes present in the drawing are listed; alternatively select the currently active class or create a new class from the Organization dialog box by clicking **Classes** (see "Classes" on page 176).
Classes	Edits class attributes
Object Attributes	Specifies the ridge attributes

3. Click **OK** to create the ridges.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Class Properties
Framing a Roof
Custom Frame Element Sizes

Creating Trimmers

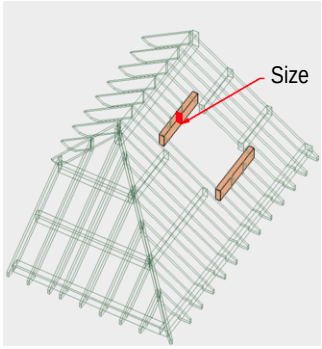
To create trimmers:

1. In the Roof Framer dialog box, click the Trimmers tab, and select **Create Trimmers**.
2. Specify the trimmer parameters.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Trimmer Size	Specifies the trimmer size in width x height, in current file units. Sizes can be edited by selecting **Edit List** (see “Custom Frame Element Sizes” on page 470).
Set top of Trimmers to	
Top of Roof object	Matches the tops of the trimmers to the top of the roof object
Bottom of Roof object	Treats the roof object as a roof deck and matches the tops of the trimmers to the bottom of the roof
Class	Select a class for the roof element to control its appearance and visibility. The classes present in the drawing are listed; alternatively select the currently active class or create a new class from the Organization dialog box by clicking **Classes** (see “Classes” on page 176).
Classes	Edits class attributes
Object Attributes	Specifies the trimmer attributes

3. Click **OK** to create the trimmers.



Setting Class Properties
Framing a Roof
Custom Frame Element Sizes

Creating Hip and Valley Rafters

To create hip and valley rafters:

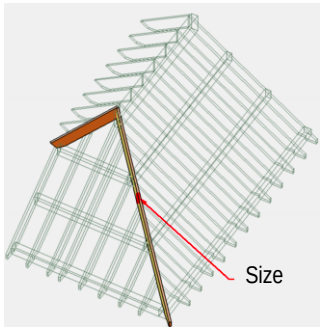
1. In the Roof Framer dialog box, click the Hips/Valleys tab, and select **Create Hips and Valleys**.
2. Specify the hip and valley rafter parameters.

Click to show/hide the parameters.

Parameter	Description
Hip/Valley Size	Specifies the size of the hip and valley rafters in width x height, in current file units. Sizes can be edited by selecting **Edit List** (see “Custom Frame Element Sizes” on page 470).
Set top of Hips and Valleys to	
Top of Roof object	Matches the tops of the hips and valleys to the top of the roof object

Parameter	Description
Bottom of Roof object	Treats the roof object as a roof deck and matches the tops of the hips and valleys to the bottom of the roof
Class	Select a class for the roof element to control its appearance and visibility. The classes present in the drawing are listed; alternatively select the currently active class or create a new class from the Organization dialog box by clicking **Classes** (see “Classes” on page 176).
Classes	Edits class attributes
Object Attributes	Specifies the hip and valley rafter attributes

3. Click **OK** to create the hip and valley rafters.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Class Properties
Framing a Roof
Custom Frame Element Sizes

Custom Frame Element Sizes

The size of the framing elements can be selected from a pre-set list, or the list can be edited to contain desired or custom sizes.

To edit the size list of the frame elements:

1. From the **Size** field in one of the framing element tabs of the Roof Framer dialog box, select **Edit List**.

   The Edit Size List dialog box opens, listing the currently available sizes of the roof element.

Click to show/hide the parameters.

Parameter	Description
Add	Adds a size to the list; specify the width and height of the new size
Remove	Deletes the currently selected size
Edit	Edits the currently selected size; enter the new width and height parameters
Move Up/Move Down	Changes the order of the sizes by moving the selected size up or down in the list

2. Click **OK** to edit the size list.

   The text files which populate the size lists in the framing element dialog boxes can also be edited. The text files are located in [Vectorworks]\Plug-Ins\VW_Arch\Data.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Framing a Roof

A Creating Framing Members

The **Framing Member** tool draws framing elements for rafters, beams, joists, and framing members with a horizontal projection.

For a file created in a version of Vectorworks software prior to 2009, board, joist, rafter, and original framing member objects are considered to be “legacy” objects. To convert them to the current functionality of the framing member tool, select **Tools > Utilities > Update Plug-in Objects**; see “Migrating from Previous Versions” on page 28.

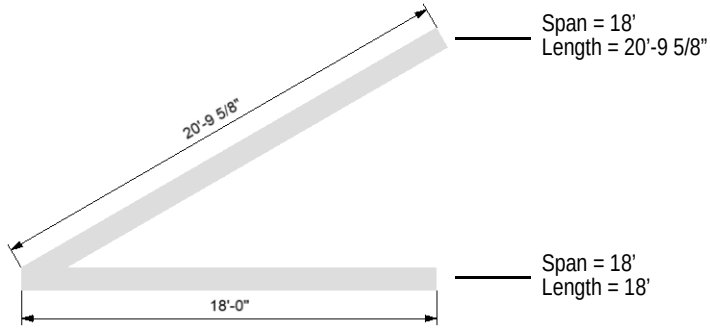
 To create a framing member:

1. Select the **Framing Member** tool from the Detailing tool set.
2. Click to begin drawing the linear framing member object. Click to set the length of the object.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.
3. The framing member object is drawn.

   The starting point and direction in which the framing member is drawn determine which end is which. When a single framing member object is selected in Top/Plan view, a blue arrow is placed at the starting point and indicates the object’s direction.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the angle of the framing member object
Span	Specifies the horizontal projection of the framing member in the floor plan; for rafters, specifies the horizontal distance from the inside surface of the supporting wall to the inside surface of the ridge board.  The **Span**, **Length**, and **Pitch** parameters are interdependent and are automatically calculated. When the span value is changed, the length is recalculated; the pitch remains fixed.
Length	Specifies the length of the framing member When the length value is changed, the pitch is recalculated; the span remains fixed.

Parameter	Description
Width	Indicates the framing member profile width
Height	Sets the framing member profile height
Type	Select the form of framing element: rafter, solid beam, wood floor truss, open-web steel joint, cold-formed steel channel, steel section, or custom profile
Structural Use	Provides a label describing how the framing member is used. To specify your own structural use name, click **Other**. The Enter Special Structural Use dialog box opens; enter text for the structural use of this framing member and click **OK**.
Volume	Displays the calculated volume of the framing member
Volume Units	Select the units for calculating the framing member volume: board feet, cubic feet, cubic meters, or the default volume units set for the file (see “Units” on page 68)
Quantity Label	Describes the standard material size required for the specified framing member, for use in quantity worksheets. When there is no standard material size that accommodates a specified measurement, the label displays a ?. You can edit the Lumber Sizes.txt file (located in [Vectorworks]\Plug-ins\VW_Arch\Data\Lumber Sizes.txt) to add material sizes that are available in your locale. Restart the Vectorworks software after editing the .txt file for the changes to take effect.
Pitch	Specifies the slope of the framing member along its span When the pitch value is changed, the length is recalculated; the span remains fixed.
Beginning/Ending miter	Specifies the miter angle at each end of the framing member. This is the angle that the framing member is cut, in plan.
Beginning/Ending bevel	Specifies the bevel angle at each end of the framing member. This is the angle that the framing member is cut, in elevation.
2D Display	Select the 2D display method for the framing member: solid, centerline, width, or width with centerline
Show Label	Displays the text entered in **Label Text** as a label next to the 2D representation of the framing member. After placement, adjust the text position with the control point. 
Label Text	Enter the text to display in 2D as a label
Vertical Reference	If you change the vertical reference from top, center, or bottom, the Change Vertical Reference dialog box opens; if the object position is maintained, the framing member remains stationary and the placement line is adjusted. If the defining line is maintained, the framing member object moves accordingly.
Rafter	
Bearing inset	Specifies the offset distance of the framing member’s bearing point (insertion point) from its end
Overhang	Specifies the additional length that the framing member extends past its end

Parameter	Description
Eave Style	Select vertical, horizontal, double, or square eave profile
Fascia Height	For double eave styles, specifies the height of the end of the framing member profile
Upper Notch Offset/ Depth	Indicates the size of an optional notch located at the top of the framing member
Solid Beam	
Draw 2D Hangers	Displays 2D graphics indicating framing hangers at start, end, both, or none
Roll Angle	Specifies the angle the framing member rotates about its cross-section axes
Wood Floor Truss	
Draw 2D Hangers	Displays 2D graphics indicating framing hangers at start, end, both, or none
Flange Height	Sets the height of the truss top and bottom chords; the chord width is set by the object width
Web Width/Height	Specifies the size of the truss web components; if **Use Solid Web** is selected, the **Web Width** parameter specifies the thickness of the solid web
Panel Spacing	Indicates the spacing of the truss “panels” (subdivisions)
Draw Verticals	Displays vertical web components between each sloping web component
Use Solid Web	Toggles the web component configuration between “open-web” and solid, “I-beam” style
Open-Web Steel Joist	
Flange Height	Sets the height of the truss top and bottom chords; the chord width is set by the object width
Web Width/Height	Specifies the size of the truss web components
Gauge	Sets the thickness of the top and bottom chords
Panel Spacing	Indicates the spacing of the truss “panels” (subdivisions)
Draw Verticals	Displays vertical web components between each sloping web component
Bottom Bearing	Toggles the configuration of the truss between top chord and bottom chord bearing
Cold-formed Steel Channel	
Draw 2D Hangers	Displays 2D graphics indicating framing hangers at start, end, both, or none
Flange Height	Specifies the length of the top and bottom flange extensions
Gauge	Sets the thickness of the framing member
Roll Angle	Specifies the angle the framing member rotates about its cross-section axes
Steel Section	
Roll Angle	Specifies the angle the framing member rotates about its cross-section axes
Select Structural Shape	Click to open the Select Structural Shape dialog box to select the structural shape to use for steel sections. Select the structural shape from the default content or the current file’s content. Select the shape’s series and size, and click **OK**.
Structural Shape data	Displays the selected structural shape symbol name, series, and size
Custom Profile	

Parameter	Description
Profile Symbol Def	Displays the selected profile symbol name
Choose Profile Symbol	For custom structural profiles, click **Choose Profile Symbol** to select a structural profile symbol from the default content or the current file's content. Select a symbol from the graphical list of **Symbols** and click **OK**. The profile name displays in the Object Info palette.
Roll Angle	Specifies the angle the framing member rotates about its cross-section axes

Resource Libraries

Framing a Floor

Use the **Create Joists from Poly** command to create the joist and perimeter objects necessary to frame a floor. If the floor is a slab object, select the slab component to serve as a boundary for the joist area. The command creates framing member objects.

To frame a floor:

1. Select a slab, polygon, polyline, or floor object.

   If a stairwell hole exists in the floor, solid beam framing member objects are used for the headers.

2. Select **AEC > Framing > Create Joists from Poly**.

   The Create Joist from Poly dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Use Slab Component	If a slab is selected, its components display in the list. Select a component to serve as the boundary for the joist area; the joists are created at the same elevation as the selected slab component.
Draw Joists	Creates joist objects for floor framing
Elevation of Top of Joist	Sets the elevation relative to the layer elevation value for the top of the joist
Center-to-Center Spacing	Specifies the spacing from the center of one joist to the center of the next joist
Class	Select a class for the joists to control their appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, or select the currently active class.
Set Joist Properties	Opens the Set Joist Properties dialog box; proceed with step 3
Set Joist Attributes	Opens the Set Joist Attributes dialog box; proceed with step 5
Draw Perimeter Framing Members	Draws framing members around the perimeter of the floor
Elevation of Top of Framing	Sets the elevation relative to the layer elevation value for the top of the perimeter framing
Class	Select a class for the framing members to control their appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, or select the currently active class.

Parameter	Description
Set Framing Member Properties	Opens the Set Framing Member Properties dialog box; proceed with step 3
Set Framing Member Attributes	Opens the Set Framing Member Attributes dialog box; proceed with step 5
End Condition of Joists at Perimeter	Select the type of condition for the ends of the joists where they connect to the perimeter framing: Inside Face, Centerline, or Outside Face
Corner Condition of Perimeter	Select the type of condition for the corners of the perimeter framing: lapped or mitered joints
Delete Source Geometry	Deletes the original source object after the framing is created

3. Select whether to draw joists and/or perimeter framing members for the floor framing. Then, specify the properties of the joists and the perimeter framing. Click **Set Joist Properties** or click **Set Framing Member Properties**.

   The Set Joist Properties or Set Framing Member Properties dialog box opens. The available parameters depend on the **Type** of joist or framing member selected.

Click to show/hide the parameters.

Parameter	Description
Width	Specifies the joist or framing member profile width
Height	Specifies the joist or framing member profile height
Type	Select the form of the joist or framing member element: rafter, solid beam, wood truss, open-web steel joist, cold-formed steel channel, steel section, or custom profile
Profile Symbol Def	For custom profiles, displays the selected profile symbol name
Choose Profile Symbol	For custom profiles, click **Choose Profile Symbol** to select a structural profile symbol from the default content or the current file's content. Select a symbol from the graphical list of **Symbols** and click **OK**. The profile name displays in the Object Info palette.
2D Display	Select the 2D display method: solid, centerline, width, or width with centerline
Show Label	Displays the text entered in **Label Text** as a label next to the 2D representation of the joist or framing member
Label Text	Enter the text to display in 2D as a label
Vertical Reference (Rafter, Solid Beam, Wood Truss, Open-Web Steel Joist, Cold-formed Steel Channel, Steel Section)	Select whether the object's defining line is at the top, center, or bottom
Draw 2D Hangers (Solid Beam, Wood Truss, Cold-formed Steel Channel)	Displays 2D graphics indicating framing hangers at start, end, both, or none

Parameter	Description
Flange Height (Wood Truss, Open-Web Steel Joist, Cold-formed Steel Channel)	Sets the height of the truss top and bottom chords; the chord width is set by the object width
Web Width/Height (Wood Truss, Open-Web Steel Joist, Cold-formed Steel Channel)	Specifies the size of the truss web components; if **Use Solid Web** is selected (wood truss only), the **Web Width** parameter specifies the thickness of the solid web
Gauge (Open-Web Steel Joist, Cold-formed Steel Channel)	Sets the thickness of the top and bottom chords
Panel Spacing (Wood Truss, Open-Web Steel Joist)	Indicates the spacing of the truss “panels” (subdivisions)
Draw Verticals (Wood Truss, Open-Web Steel Joist)	Displays vertical web components between each sloping web component
Bottom Bearing (Open-Web Steel Joist)	Toggles the configuration of the truss between top chord and bottom chord bearing
Use Solid Web (Wood Truss)	Toggles the web component configuration between “open-web” and solid, “I-beam” style
Section Type (Steel Section)	Select the type of structural shape for the steel section: Angle, Bulb Flat, Channel, I-beam, Rectangular Tubing, Round Tubing, Square Tubing, Tee, or Wide Flange
Section Series (Steel Section)	Select the series type for the steel section
Section Shape (Steel Section)	Select the size of the steel section

4. Click **OK** to set the joist or perimeter framing member properties.
5. Specify the attributes of the joists and the perimeter framing. Click **Set Joist Attributes** or click **Set Framing Member Attributes**.

   The Set Joist Attributes or Set Framing Member Attributes dialog box opens. The dialog boxes are identical whether setting joist or perimeter framing attributes, and contain a Graphic Attributes tab and Texture tab (Renderworks required).

Click to show/hide the parameters.

Parameter	Description
Graphic Attributes	Each graphic attribute can be set by the class style or to specific settings. If set by the class style, the attributes are controlled by the class selected in the Create Joists from Poly dialog box.
Fill	Select Class Style to use the class style for the fill attribute, choose None to apply no fill, or choose a fill style

Parameter	Description
Pen	Select Class Style to use the class style for the pen attribute, choose None to apply no pen, or choose a solid/dash or pattern style
Line Thickness	Select Class Thickness to use the class style for the line thickness attribute, choose Set Thickness to specify the line thickness, or select a line thickness
Texture (Renderworks required)	Select Class to use the class texture for the framing element (see "Applying Object Textures by Class" on page 1542), choose None to apply no texture, or choose Select to set the texture parameters
Texture	Select the texture to apply from either the default content or the current file's content
Map Type	Select the texture map type
Radius	For sphere and cylinder maps, sets the texture radius; the default radius is the same as the 3D object radius. Increasing this value reduces the size of the texture on the object.
Start/End Cap	Applies the texture to the start and end caps of the framing element
Repeat Horizontally/ Vertically	Repeats the texture in a horizontal and/or vertical direction
Scale Factor	Determines the texture size when projected onto the object; either enter a scale value or use the slider to change the scale
Horizontal/Vertical Offset	Sets the start location of the texture horizontally and vertically
Rotation	Sets the angle of texture rotation; either enter a rotation value from 0 to 360 degrees or use the slider to change the rotation angle

6. Click **OK** to set the joist or perimeter framing member attributes.
7. Click **OK** to close the Create Joist From Poly dialog box.
8. If joists are to be created, an alert dialog box requests the selection of two points to set the joist orientation. Click **OK** to close the dialog box and click two points to indicate the desired joist orientation.
9. The joists and/or perimeter framing member objects are created. In addition, a Joist Take-Off worksheet is automatically created and added to the Resource Browser.

   The framing member parameters can be edited in the Object Info palette. These parameters are identical to those of a single framing member object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
The Attributes Palette
Creating Framing Members

A Framing a Wall

The Vectorworks Architect product creates a highly detailed estimate of the placement and number of studs needed to frame walls. In addition to showing stud placement in a framing diagram, the **Wall Framer** command also automatically generates frame elevation drawings and two different worksheets, Frame TakeOff and Frame Wall Info.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To use the wall framer:

1. Ensure that the walls have the desired height. If necessary, change the wall height.
2. Select **AEC > Framing > Wall Framer**.

   If a framing model has not yet been created, the New Framing Model dialog box opens. Enter a name for the framing model design layer (up to eight characters).
3. Click **Create**.

   The Wall Framing dialog box opens. Select the desired settings for the framing model.

Click to show/hide the parameters.

Parameter	Description
Model Name	Select the framing model layer to be used
New	Opens the New Framing Model dialog box to create a new model
Delete	Deletes the currently selected model. Click **Yes** to delete the selected model. This operation cannot be undone. This command cannot be used if there is only one model in the document.
Framer Output	
3D Model	Creates a 3D model layer, complete with the placement of studs, top plates, sole plates, and other components required for framing
2D Model	Creates framing plan and framing diagram layers. The framing plan is a top view of the wall showing the sole plate and stud placement and is drawn in a layer that parallels the layers in the framing model. The top plates are removed to show the stud placement. The framing diagram is an elevation view showing the sole plate, top plates, and stud placement. The framing diagram is drawn in a new design layer and generates as many layers as needed.
Worksheets	Creates two sets of worksheets. The first set provides details on the framing results, including a list of studs sorted by layer, class, frame, and size (Frame TakeOff). The second set provides a summary of frame information, sorted similarly, but providing area and linear footages itemized for each framed wall (Frame Wall Info).
Frame the following layers	Select which layer(s) contain wall data to frame
Options	Opens the Wall Class dialog box to allow the editing of wall class values; see step 4

   The framing diagram and the 3D model representations also include symbols and inserted items, such as doors and windows.

4. Click **Options**.

   The Wall Class dialog box opens. Enter the framing parameters for each wall class.

Click to show/hide the parameters.

Parameter	Description
Wall Class	
Name	Lists the classes that contain walls to be framed; walls in the classes in this list are included in the framing model. The framing parameters for each class are displayed beneath.
New	Click to add an additional wall class to the list to be framed
Delete	Deletes the currently selected wall class from the list to be framed. Click **Yes** to delete the selected class. This operation cannot be undone.
Stud Spacing	Enter the center-to-center distance between studs in this wall class
Sheet Stock Width	Enter the width of the wall sheathing; this is the horizontal sheathing dimension
Max Plate Length	Enter the length of lumber to be used for top and bottom frame members in this wall class
Sheet Stock Height	Enter the width of the wall sheathing; this is the vertical sheathing dimension
Lumber	
Nominal	Select the lumber type to use for this wall class; the default types are 2” x 4”, 2” x 6”, and 2” x 10”
New	Click to create a new nominal lumber type; enter the name and click **OK**; then, with the new type selected in the **Nominal** list, enter the lumber dimensions in **Size**
Delete	Deletes the currently selected nominal lumber type. Click **Yes** to delete the selected lumber type. This operation cannot be undone. This command cannot be used if there is only one lumber type in the document.
Size	Enter the true short dimension for the lumber
by	Enter the true long dimension for the lumber
Output Options	
Fire Blocking	Select to have extra blocking added to this wall class between studs on walls that are shorter than the sheet stock height
Double End Studs	Select to have two studs, rather than one stud, placed at each wall end in this wall class
Double Top Plate	Select to have two plates, rather than one plate, placed at top of each wall in this wall class
Double Bottom Plate	Select to have two plates, rather than one plate, placed at the bottom of each wall in this wall class

5. Click **OK** to return to the Wall Framing dialog box.
6. Click **OK** to create the framing model.

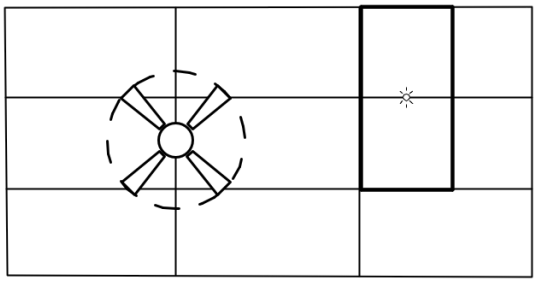
   The program creates the estimated framing for the walls and any other output information requested.

   The 2D results display in Top/Plan view without a top plate. To view the results with a top plate, switch to another view such as Front, Back, Left, or Right.

   To view the 3D results, switch to a view such as Left Isometric or Right Isometric.

   The Framer TakeOff and Framer Wall Info worksheets, if created, display in the drawing area.

Inserting Ceiling Grid Objects



Use the **Ceiling Grid** tool to insert a ceiling grid object containing tiles with user-specified length, width, and placement angle. A ceiling grid object can also be created by drawing a polyline and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

To insert a ceiling grid object:

1. Click the **Ceiling Grid** tool from the Furn/Fixtures tool set or the MEP tool set.
2. Click in the drawing file to set the start point of the ceiling grid object, and then click to set the position of each additional vertex. Click the last point to complete creation of the ceiling grid object. If the polyline is open, the program automatically completes it.

   The Ceiling Grid object properties dialog box opens. Click **OK** to accept the default settings.
3. Use the **Reshape** tool to modify the locations of the vertices after object creation, or use the vertex editing controls on the Object Info palette to move the vertices or change the degree of vertices. The tiles are automatically adjusted to fit the new shape.

Click to show/hide the parameters.

Parameter	Description
Angle	Specify the tile placement angle
Tile Width	Specify the tile width
Tile Length	Specify the tile length
Vertex parameters	Edits the ceiling grid object path vertices; see “Editing Vertex-Based Objects” on page 1002

For information on editing object vertices, see “Reshaping Objects” on page 1043.

Floors and Slabs

Creating Floors

The **Floor** command can be used to create not only floors, but also decks, patios, stages, platforms, and lofts. The command converts any 2D drawing object into a hybrid (2D/3D) object of any shape and thickness. The source object must be flat and must also be parallel to the active layer plane.

To create a floor:

1. Select the 2D object to turn into a floor.
2. Select the **Floor** command from the appropriate menu:
   - Fundamentals workspace: **Model > AEC > Floor**
   - Architect workspace: **AEC** > **Floor**
   - Landmark workspace: **Landmark > Architectural > Floor**
   - Spotlight workspace: **Spotlight > Architectural > Floor**

   The Create Floor dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Bottom Z	Specifies the distance above the active layer plane that the floor will begin (the Bottom Z height). In most cases, use the default of 0, which starts the floor on the active layer plane.
Thickness	Sets the floor thickness

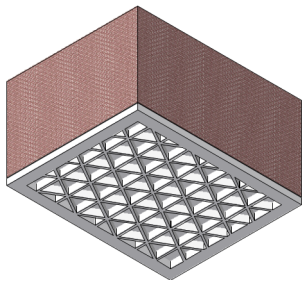
3. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

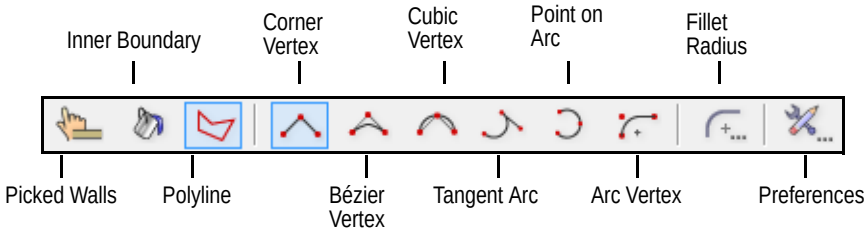
Framing a Floor

Creating Slabs

The **Slab** tool creates architectural floor slabs. Slabs can be drawn manually or they can be based on, and associated with, an existing set of walls. The method of associating slab components with wall components can be specified, to create accurate and detailed cross-section views.



To draw a free-standing slab manually with the **Slab** tool, select the Polyline mode. If the walls are already drawn, select the Picked Walls mode or Inner Boundary mode to create a slab bounded by visible walls.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mode	Description
Picked Walls	Defines the perimeter of a slab based on a closed set of selected walls
Inner Boundary	Creates a slab object in a clicked area that is bounded by visible walls
Polyline	Draws a polyline slab object; as with a polyline, select one of six types of control points for the vertices from the Tool bar
Corner Vertex	For Polyline mode, creates the slab using polyline segments with straight lines and angled vertices at the control points
Bézier Vertex	For Polyline mode, creates the slab using polyline segments with curves pulled toward, but not touching the control points
Cubic Vertex	For Polyline mode, creates the slab using polyline segments with curves that pass through the control points
Tangent Arc	For Polyline mode, creates the slab using polyline segments that are tangent to the previous segment
Point on Arc	For Polyline mode, creates the slab using polyline segments that are drawn by clicking three points: the start point, a point the arc passes through, and the end point
Arc Vertex	For Polyline mode, creates the slab using polyline segments with curves that look like a fillet placed at the control points
Fillet Radius	For Polyline mode, sets the fillet radius when the Arc Vertex mode is selected
Preferences	Sets the default parameters that are used for each slab object

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Drawing Slabs Manually
Creating Automatically Bounded Slabs from Existing Walls
Converting Slab Boundary Associations
Using Slab Styles
Creating Slab Components
Editing Slabs
Creating Floors

Drawing Slabs Manually

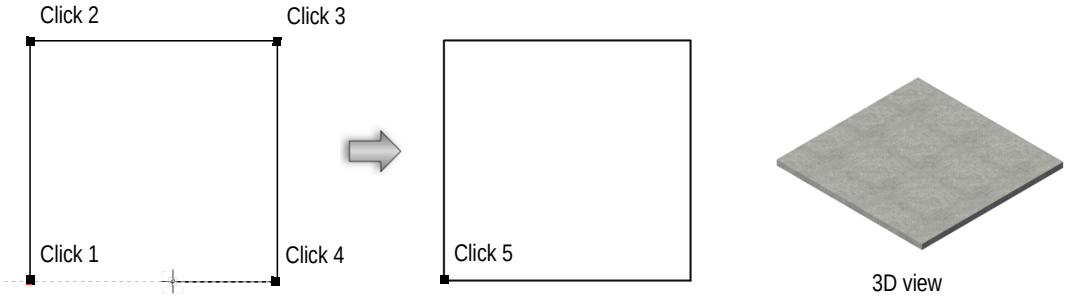
To draw a manual slab object that is not associated with any walls:

1. Click the **Slab** tool from the Building Shell tool set, and select **Polyline** mode from the Tool bar.
2. To draw an unstyled slab, select <Unstyled> from the Slab Style list on the Tool bar. To create a styled slab, select it from the Tool bar **Slab Styles** list, which displays the slab style resources in this file as well as default slab styles. Create a new slab style by clicking **Preferences** from the Tool bar; see “Creating Slab Styles” on page 486.
3. Select a polyline drawing mode. See “Polyline Tool” on page 298.

   A slab can also be created by drawing a polyline and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).
4. Similarly to drawing a polygon, create a closed slab shape.

   Double-click while drawing to close the slab automatically.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Every slab contains a component designated as the datum component. The top of this component is the Z-reference datum, and is fixed vertically (it does not change with any changes to the slab thickness or definition). The reference datum of the slab can be set to the active layer elevation value, or set to be bound by one of the story levels defined for the story or the story below it. By setting the slab datum to a level type, if the elevation of the associated story changes, the datum of the slab changes automatically to match.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Automatically Bounded Slabs from Existing Walls
Converting Slab Boundary Associations
Using Slab Styles
Creating Slab Components
Editing Slabs
Creating Slabs

Creating Automatically Bounded Slabs from Existing Walls

Slabs can be automatically created by first selecting the enclosing walls or by clicking within the boundary of a closed set of visible walls. The slabs are automatically associated with their bounding walls.

Use classes to distinguish interior and exterior walls to easily place slabs associated with the correct wall set.

Every slab contains a component designated as the datum component. The top of this component is the Z-reference datum, and is fixed vertically (it does not change with any changes to the slab thickness or definition). The reference datum of the slab can be set to the active layer elevation value, or set to be bound by one of the story levels defined for the story or the story below it. By setting the slab datum to a level type, if the elevation of the associated story changes, the datum of the slab changes automatically to match.

The slab edge path is defined by the outside or inside face of the walls, or by components within the wall, as defined by the slab component properties. If the walls are edited (whether moved, reshaped, or replaced, or due to a change in the wall style, the wall thickness, or the wall components), the slab edge path updates automatically. However, if the walls are edited so significantly that the slab cannot retain its association, you have the option to create a manual slab from the automatically bounded slab.

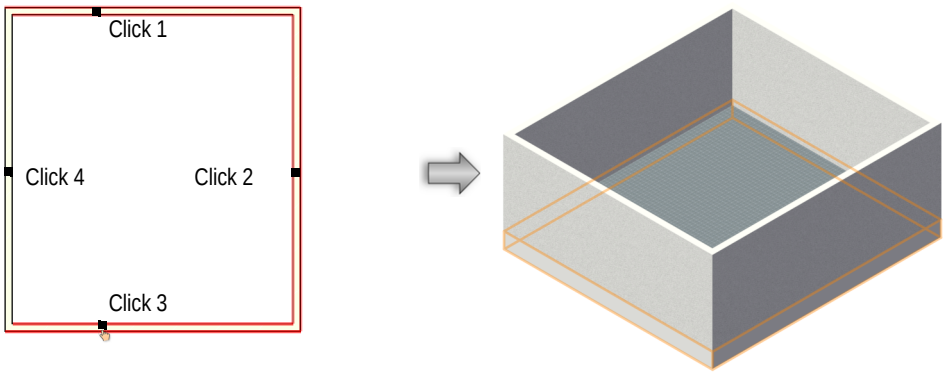
Because the walls and slab are associated, section views display accurately. When creating a section view, similar component fills can be merged to display as a single unit (see “Advanced Sheet Layer Viewport Properties” on page 1642).

Once an automatically bounded slab is created, it can be moved to another layer and still retain its association with the walls.

To create an automatically bounded slab within a selected set of visible walls:

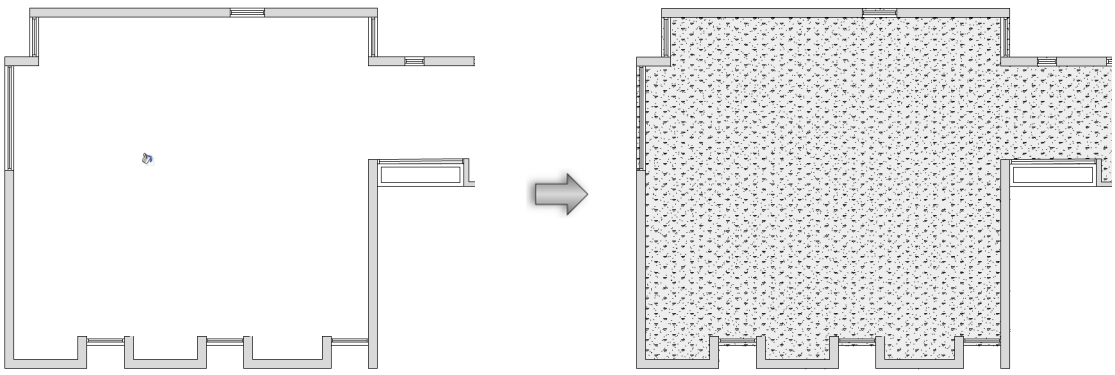
1. Click the **Slab** tool from the Building Shell tool set, and select **Picked Walls** mode from the Tool bar.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. To draw an unstyled slab, select <Unstyled> from the Slab Style list on the Tool bar. To create a styled slab, select it from the Tool bar **Slab Styles** list, which displays the slab style resources in this file as well as default slab styles. Create a new slab style by clicking **Preferences** from the Tool bar.
3. Click on each wall that forms the wall set and press Enter, or click the check mark button on the Tool bar.
4. The slab is created.



To create a slab within a closed set of visible walls:

1. Click the **Slab** tool from the Building Shell tool set, and select **Inner Boundary** mode from the Tool bar.
2. To draw an unstyled slab, select <Unstyled> from the Slab Style list on the Tool bar. To create a styled slab, select it from the Tool bar **Slab Styles** list, which displays the slab style resources in this file as well as default slab styles. Create a new slab style by clicking **Preferences** from the Tool bar.
3. Click in an open area of the drawing that is bounded by walls; the walls must be visible, joined together, and on layers that have the same layer scale as the slab layer.
4. The slab is created.



With exterior walls visible, click to place a concrete-style slab

Send the created slab to the back (**Modify** > **Send** > **Send to Back**) for Top/Plan view

Click here for a video tip about this topic (Internet access required).

Drawing Slabs Manually
Converting Slab Boundary Associations
Creating Slab Styles
Using Slab Styles
Creating Slab Components
Editing Slabs

Creating Slabs

Converting Slab Boundary Associations

Converting to Automatically Bounded Mode

Manually drawn slabs can be associated with a set of walls, converting them to automatically bounded mode. Similarly, an automatically bounded slab can be associated with a new set of walls by this method.

To convert manual slabs to automatically bounded slabs:

1. Select the manual slab. In the Object Info palette, the **Bounding** mode displays Manual.
2. Click Pick Boundary from the **Bounding** list.

   The **Slab** tool is automatically activated, and Inner Boundary mode is selected (Picked Walls mode can also be selected).
3. For Inner Boundary mode, click in an open area of the drawing that is bounded by walls. For Picked Walls mode, click on each wall that forms the wall set and press Enter, or click the check mark button on the Tool bar.
4. The manual slab is associated with the wall set. Its location and size may change as it is bounded by the wall set.

Converting to Manual Mode

Automatically bounded slabs can be converted to manual mode, releasing them from their association with walls.

To convert automatically bounded slabs to manual slabs:

1. Select the automatically bounded slab. In the Object Info palette, the **Bounding** mode displays Auto-Bounded.
2. Click Manual from the **Bounding** list.
3. The slab is disassociated from its wall set, though it remains in place.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Drawing Slabs Manually
Creating Automatically Bounded Slabs from Existing Walls
Using Slab Styles
Creating Slab Components
Editing Slabs
Creating Slabs

Using Slab Styles

Unstyled slabs can be created; in a new file, the default unstyled slab includes a single component. The attributes of an unstyled slab can be changed from the Attributes palette. The use of slab styles, however, facilitates drawing slabs by saving the slab preferences settings so that they can be easily applied to other slabs. Slab styles are resources that can be imported into other files and shared as office standards. Slab styles, textures, and hatches are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser). Setting and saving the dynamic height information of a slab in its slab style is a convenient way of automatically setting the defined height condition when using slabs with stories.

If a selected slab style's parameters are edited, the slab style automatically changes to unstyled, indicating that the edited style must be saved as a new style to save changes (alternatively, the changes can be applied as an unstyled slab, and not get saved).

Unused slab styles can be purged; see "Purging Items from a File" on page 1011.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries

Creating Slab Styles

To create a slab style:

1. Click the **Slab** tool from the Building Shell tool set, and click **Preferences** from the Tool bar.

   The Slab Preferences dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Slab Style	To modify an existing slab style, select the **Slab Style** from either the default content or the current file's content. Alternatively, select Unstyled as the slab style and set the parameters.
Save Preferences as Slab Style	Saves the current preference settings as a new slab style. The Assign Name dialog box opens; enter a slab style name and click **OK**
Preview	Displays a sectioned preview of the slab structure, including any defined components. The top of the preview indicates the top part of the slab as it will be drawn.
Overall Thickness	Displays the thickness of the slab. The slab thickness is always determined by the thickness of its components; a slab always has one component by default.
Edit Slab Attributes	Opens the Slab Attributes dialog box, to specify the slab fill, pen, and opacity settings
Components	Lists the components that form the structure of the slab, in order from top to bottom as displayed in the preview. To change the order of a component, click and drag within the # column.
Datum	One of the slab components is designated as the datum component by clicking in the Datum column. A check mark indicates that the component is the datum component. The top of this component is the Z-reference datum, and is fixed vertically (it does not change with any changes to the slab thickness or definition)
Other Component settings	The remaining columns in the component list display the component thickness, edge offset method, and class setting
New	Opens the Slab Component Settings dialog box to define the components of the slab; see "Creating Slab Components" on page 491 for information on creating components
Edit	Opens the Slab Component Settings dialog box to edit the selected component's thickness and attributes (alternatively, double-click on a component to open the Slab Component Settings dialog box)
Duplicate	Duplicates the selected slab component; the duplicate is added to the component list right below the originally selected component
Delete	Deletes the selected slab component; the slab thickness is adjusted accordingly

2. Click **Edit Slab Attributes** to define the slab attributes in Top/Plan view.

The Slab Attributes dialog box opens. The slab attributes of an unstyled slab are initially set to the parameters displayed in the Attributes palette. If they are changed here, the Attributes palette reflects the changed attributes of the selected slab (after exiting the Slab Preferences dialog box).

Fill, pen, and opacity can be set by class rather than by the attributes in the Slab Attributes dialog box. If the slab class is changed later, the slab changes to use the attributes of the new class. Slab attributes cannot be overridden on a per-instance basis; if a slab style uses class attributes, all slabs of that style must use class attributes. However, slabs of the same slab style can be placed in different classes.

Click to show/hide the parameters.

Parameter	Description
Fill	
Style	Specify the slab fill attributes. Select None for no fill, or Class Style to set the slab fill attributes by class rather than by the parameters in the Slab Attributes dialog box. Selected hatch, gradient, or image fills, if not already present in the file, are imported and added to the Resource Browser (default content is automatically imported into the current file at the point of use and displays in the Resource Browser).
Pen	
Style	Specify the slab pen attributes. Select None for no pen, Line Type to select a complex line type, or Class Style to set pen attributes by class rather than by the parameters in the Slab Attributes dialog box.
Thickness	When a pen style has been selected, specify the line thickness. If Line Type is selected as the pen style, also select the line type to use.
Opacity	
Use Class Opacity	Sets slab opacity by class rather than by the parameters in the Slab Attributes dialog box
Opacity	Specifies the transparency of the slab; drag the slider to the left to increase the transparency, or enter a percentage directly in the box to the right of the slider

3. Click **OK** to return to the Slab Preferences dialog box.
4. Click the Insertion Options tab to set the slab insertion options.

Click to show/hide the parameters.

Parameter	Description
Datum LayerZRef	Sets the slab's datum height to be defined by the layer elevation, or set it to be bound by one of the story levels defined for the story or the story below it. By setting the slab datum to a level type, if the elevation of the associated story changes, the datum of the slab changes automatically to match.
Datum Offset from Layer Z	Specifies the offset (positive or negative) of the slab's datum component from the layer elevation (see "Setting Design Layer Properties" on page 165)
Class	Specifies the default class for the slab

5. Click the Textures tab to select textures for the slab parts (Renderworks required).

Click to show/hide the parameters.

Parameter	Description
Use Component Textures	Uses the textures defined for the slab components to texture the slab This option can also be selected for an existing slab from the Render tab of the Object Info palette.
Use Object Textures	Uses the textures defined below for the slab parts (see "Applying a Texture to an Object" on page 1530 for information on applying textures to an object with parts)
Part	Applies the texture selected in **Texture** to the selected parts of the slab
Revert to Overall	If a part was assigned a texture but it should inherit its texture from Overall instead, select the part and click **Revert to Overall**. The part moves back below the divider, and (from Overall) displays as its texture name.
Texture	Applies the selected texture to the selected **Part**. Textures can also be set from the Render tab of the Object Info palette. Textures applied from the Object Info palette override the textures set here.
No Texture	Does not apply a texture to the selected **Part**
Class Texture	Sets the slab to use the texture specified by the slab's class. The class texture is set from the Other tab in the Edit Classes dialog box. Any slabs with that slab style use class textures for that part of the slab (unless overridden). Class Texture can also be chosen for a selected slab in the Render tab of the Object Info palette.
Choose Texture	Choose a texture for the selected **Part** from the default content or the current file's content (see "Resource Libraries" on page 219). Textures set here override the object class textures.

6. Click the Data tab to specify slab record information, which is IFC-compliant and can be included in a slab style schedule. These fields are optional; enter text only where desired.
7. Click **Save Preferences as Slab Style**.

   The Assign Name dialog box opens.
8. Enter a unique name for the slab style and click **OK**.

   If the slab style name already exists, you are prompted to cancel and select a different name, or replace existing slabs with the slab style applied with the edited slab style. If you are replacing slab styles, the Slab Replacement dialog box opens; specify the slab replacement properties.
9. The new slab style is saved with the file and is listed under Slab Styles in the Resource Browser as well as the Slab Style list in the Tool bar.

   A new slab style can also be created by clicking **Resources > New Resource > Slab Style** in the Resource Browser. A slab style created in this way is not associated with the current slab preference setting, but can be applied later.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
The Attributes Palette
Applying a Texture to an Object
Setting Class Properties
Creating Slabs
Editing Slab Styles
Applying Slab Styles
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Replacing Slab Styles
Editing Slabs

Editing Slab Styles

To edit a slab style:

1. Select the slab style from the Resource Browser and click **Resources > Edit**.

   The Edit Slab Style dialog box opens.

2. Edit the slab and component parameters.

   If a new slab style **Name** is specified, it replaces the selected slab style name (similar to selecting **Rename** from the Resources menu).

3. Click **OK**.

4. If slabs with that style already exist in the drawing, the Slab Replacement dialog box opens.

   The slab style to apply cannot be selected (the edited style is applied).

5. Click **OK** to edit the slab style.

   Changes apply to any existing slabs in the drawing with the edited slab style, and will be used for any subsequent slabs created with that slab style.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Slabs
Creating Slab Styles
Using Slab Styles
Applying Slab Styles
Replacing Slab Styles
Creating Slab Components
Editing Slabs

Applying Slab Styles

By a variety of methods, slab styles can be selected for a slab before drawing it, or applied to existing slabs. Once a slab style has been selected for a slab, the Attributes palette is no longer available for changing slab attributes; attributes are set as part of the style.

To apply a slab style prior to creating the slab:

1. Select the **Slab** tool from the Building Shell tool set.
2. Select the desired style from the **Slab Style** list on the Tool bar. The available slab styles are from either the default content or the current file's content.

   Alternatively, click **Preferences** from the Tool bar. In the Slab Preferences dialog box, select the **Slab Style**.

To apply a slab style from the Resource Browser prior to drawing the slab:

1. Ensure that no slabs are selected.
2. Select a slab style in the Resource Browser, and click **Resources > Apply**.

   Alternatively, double-click on the slab style in the Resource Browser.

3. The **Slab** tool is automatically made active and the selected slab style is applied to the slab as it is drawn.

To apply a slab style to an existing slab from the Object Info palette:

1. Select one or more slabs.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. From the Object Info palette, select the slab **Style**. To apply a different (default) style, select **Replace**.

To apply a slab style to an existing slab from the Resource Browser:

1. Select one or more slabs.
2. Select the slab style from the Resource Browser and click **Resources > Apply**. Alternatively, drag the selected slab style from the Resource browser to the slab and click on the slab selection.

   The Slab Replacement dialog box opens. The slab style list is disabled (the style selected in the Resource Browser is applied).
3. Click **OK**.

A styled slab can be converted to an unstyled slab.

To remove a slab style:

1. Select one or more slabs to un-style.
2. From the Object Info palette, select **Convert to Unstyled Slab**.

   The slab is released from its style; its attributes can be edited from the Attributes palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
Creating Slabs
Creating Slab Styles
Editing Slab Styles
Replacing Slab Styles
Creating Slab Components
Editing Slabs

Replacing Slab Styles

Slab styles applied to existing slabs can be replaced with a different slab style.

A slab replacement situation may also occur if pasting a styled slab from another file.

To replace a slab style:

1. Select one or more slabs.
2. From the Object Info palette, select Replace from the **Style** list.

   The Slab Replacement dialog box opens. Specify the new slab style. The red line indicates the datum component of each slab style; replacement is aligned to the datum.

Click to show/hide the parameters.

Parameter	Description
Slab style list	From the list, select the slab style that will replace the current style
Previews	The left preview shows the current slab style, and the right preview shows the replacement slab style selected in the slab style list. Preview slabs are drawn from top to bottom, so the top of the preview indicates the top part of the slab as it will be drawn.
Replacement options	Select whether to replace the current settings for slab height, class designations, and texture assignments (Renderworks required for textures) with those of the new slab

If slabs with different slab styles were selected for replacement, the current slab preview is blank and components do not display.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

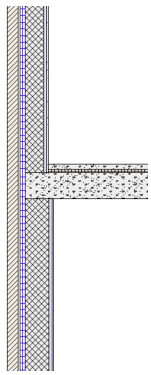
3. Click **OK** to replace the slab style of the selected slab(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Slabs
Creating Slab Styles
Editing Slab Styles
Applying Slab Styles
Creating Slab Components
Editing Slabs

Creating Slab Components

Slab components define the sections that make up a slab. For example, to indicate that a slab is made up of gypsum board ceiling, wood framing, and a plywood deck, define a component for each of these items to illustrate their location. Slab components can be offset from the edges of the walls (automatically bounded slabs) or the edge of the slab (manual slabs); their appearance can be specified for cross-section views, and they can be textured (Renderworks required), creating realistic section views and rendered views, as well as accurate slab material estimates. The area and volume of slab components (minus any holes cut) can be calculated in worksheets; see “Worksheet Functions” on page 1344).



The overall thickness of a slab is equal to the sum of its components. Component fill and pen style are only displayed in section viewports.

To define slab components prior to drawing the slab:

1. Click the **Slab** tool from the Building Shell tool set, and click **Preferences** from the Tool bar.

   The Slab Preferences dialog box opens.

2. On the Definition tab, click **New**.

   The Slab Component Settings dialog box opens. Specify the component name, thickness, edge offset, and appearance in section views and rendered views.

Click to show/hide the parameters.

Parameter	Description
Definition	
Name	Provide a name for the component, which displays in the **Components** list in the Slab Preferences dialog box
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Class	Select a class for the component to control its appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting **New**, or select the class named <Object Class>, which places the component in the same class as the slab object.
Thickness	Specifies the component's thickness; the thickness of a slab is the sum of its components. A component must have a thickness greater than 0.
Edge Offset	The method of offsetting the component from the edge of the slab depends on whether the slab is an automatically bounded slab, associated with a set of walls, or a slab drawn manually. The edge offset setting displays for each component in the component list on the Definition tab of the Slab Preferences dialog box.
Auto-Bound	Sets the component edge to the inner or outer face of the wall, outer face of the inner wall component, inner face of the outer wall component, or inner face/outer face/center of the wall component designated as the core component
Manual	Specifies the offset of the component from the edge of the slab
Texture (Renderworks required)	
No Texture	Does not apply a texture to the component
Class Texture	Sets the component to use the texture specified by the component's class. The class texture is set from the Other tab in the Edit Classes dialog box; see "Setting Class Properties" on page 179.
Choose Texture	Select a texture for the component from the default content or the current file's content (see "Resource Libraries" on page 219).
Section Fill	Specifies the component appearance in section views
Style	Specify the slab fill attributes. Select None for no fill, or Class Style to set component fill attributes by class rather than by the parameters in the Slab Component Settings dialog box. Selected hatch, gradient, or image fills, if not already present in the file, are imported and added to the Resource Browser (default content is automatically imported into the current file at the point of use and displays in the Resource Browser). See "Resource Libraries" on page 219 and "Fill Attributes" on page 1098. Use a tile fill with **Fit to Wall** selected, to represent insulation fill. See "Defining Tiles" on page 1112.
Section Pen	Specifies the component appearance in section views
Style	Specify the slab pen attributes. Select None for no pen, Line Type to select a complex line type, or Class Style to set component pen attributes by class rather than by the parameters in the Slab Component Settings dialog box. Selected line types, if not already present in the file, are imported and added to the Resource Browser (default content is automatically imported into the current file at the point of use and displays in the Resource Browser). See "Resource Libraries" on page 219 and "Pen Attributes" on page 1100.
Thickness	When a pen style has been selected, specify the line thickness; see "Line Thickness Attributes" on page 1101. If Line Type is selected as the pen style, also select the line type to use.

3. Click **OK** to create the component and return to the Slab Preferences dialog box.

The slab's **Overall Thickness** value changes to be determined by its components. As components are defined, they display in the preview. Click and drag a component in the # column to change its order.

4. To save the component settings with the slab settings as a slab style, click **Save Preferences as Slab Style**.
5. Click **OK**.

   Once created, the components of a selected slab can be edited by clicking Components from the Object Info palette.

Defining Components for Existing Slabs

To define or edit slab components for an existing, unstyled slab:

1. Select the slab(s).
2. From the Object Info palette, click **Components**.

   The Components dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Preview	Displays a preview of the slab structure, including the defined components
Overall Thickness	The thickness of a slab with components is defined by the sum of the component thicknesses
Components	Lists the components that form the structure of the slab, in order from top to bottom as displayed in the preview. To change the order of a component, click and drag within the # column.
Datum	One of the slab components is designated as the datum component by clicking in the Datum column. A check mark indicates that the component is the datum component. The top of this component is the Z-reference datum, and is fixed vertically (it does not change with any changes to the slab thickness or definition)
New	Opens the Slab Component Settings dialog box to define the components of the slab as described previously in this section
Edit	Opens the Slab Component Settings dialog box to edit the selected component's thickness and attributes (you can also double-click on a component to open the Slab Component Settings dialog box)
Duplicate	Duplicates the selected slab component; the duplicate is added to the component list right below the originally selected component; the slab thickness is adjusted accordingly
Delete	Deletes the selected slab component; the slab thickness is adjusted accordingly

3. Click **New** or **Edit**, and define or edit the components as described in "Creating Slab Components" on page 491.
4. Click **OK** to return to the Components dialog box.
5. Click **OK**. The new component definition is applied to the selected slab(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
The Attributes Palette
Defining Tiles
Setting Class Properties
Creating Slabs
Creating Slab Styles
Using Slab Styles
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Slabs

Editing Slabs

Slabs are path-based objects and can be reshaped, added to, and clipped. For example, a slab may need a hole clipped to accommodate a flight of stairs, or may need area added to represent a balcony.

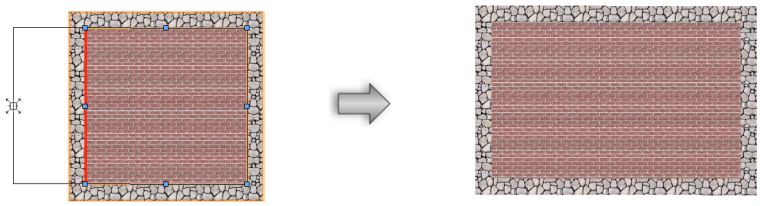
Reshaping Slabs

To reshape a slab:

1. Click the **Reshape** tool from the Basic palette.

   Alternatively, double-click on the slab to activate the **Reshape** tool.
2. Click one of the modes from the Tool bar.

   See “Reshaping Objects” on page 1043.
3. The datum component is selected for reshaping; any editing applies to the entire slab.

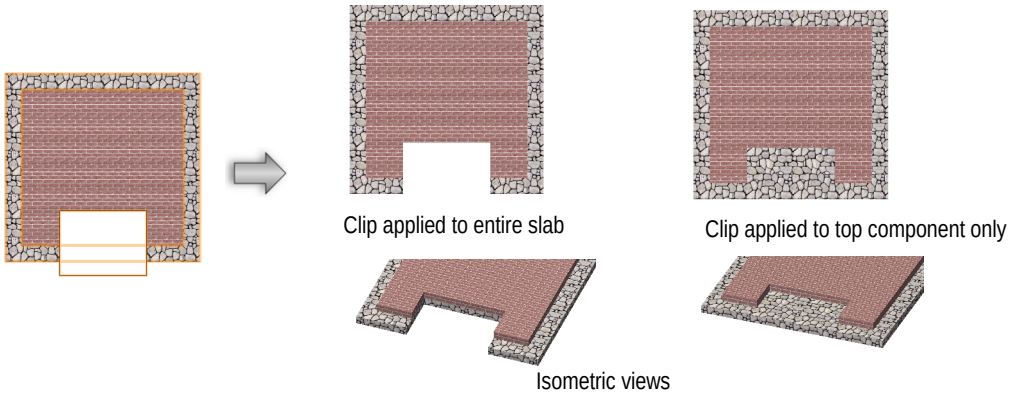


Clipping a Slab

When clipping a slab, select whether the clip operation should affect the entire slab or only certain components.

To clip a slab:

1. Create a closed 2D object to represent the clipping object. The object must overlap the slab.
2. Select both the slab and the clipping object.
3. Select **Modify > Clip Surface**.
4. The Clip shape from slab dialog box opens. Select whether to clip the entire slab, or choose the components to be clipped by selecting them.
5. Click **OK**.

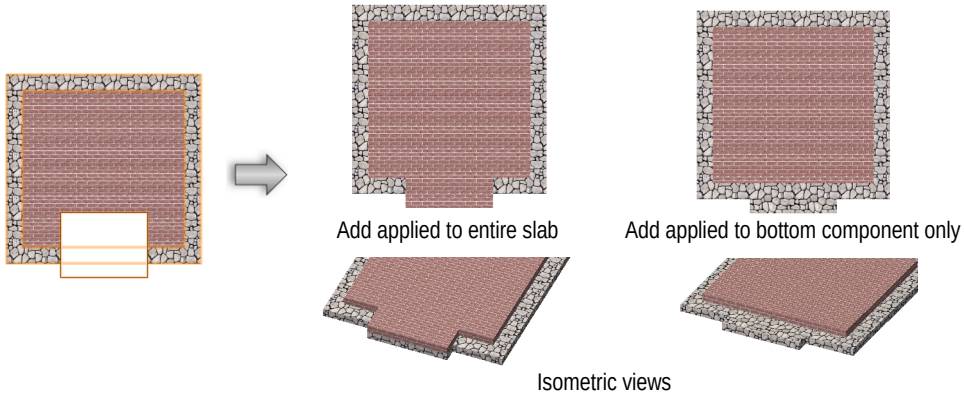


Adding to the Slab Surface

When adding to a slab, select whether the add operation should affect the entire slab or only certain components.

To add to a slab:

1. Create a closed 2D object to represent the add surface. The object must overlap the slab.
2. Select both the slab and the add surface object.
3. Select **Modify > Add Surface**.
4. The Add Shape to Slab dialog box opens. Select whether to add to the entire slab, or choose the components to be added to by selecting them.
5. Click **OK**.



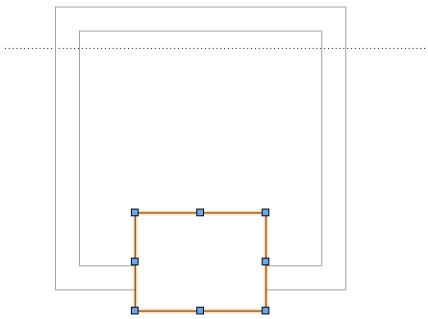
Modifying the Clipping/Adding Shape

The original editing object used to clip or add to the slab can be modified.

To modify the editing object:

1. Select the slab.
2. Select **Modify > Edit Slab.**

   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Slab** command becomes available from the **Modify** menu, and the **Exit Slab** button is visible in the top right corner of the drawing window. The editing object is selected.



3. Move, delete, or reshape the editing object (new editing objects cannot be added, however). The slab is locked and cannot be edited.
4. Click **Exit Slab** to return to the drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Walls

All Vectorworks products have tools and commands to design straight and round walls. Some of the tools and commands in the Vectorworks Design Series products are similar to those available in the Vectorworks Fundamentals product, but with extended capabilities required for architectural design. Walls can be drawn using the **Wall** tool and **Round Wall** tool, or created automatically from objects that exist in the drawing, including spaces, polygons, and shapes. The Vectorworks Architect and Landmark products also include the ability to create curtain walls.

Vectorworks Fundamentals	Vectorworks Architect, Landmark, or Spotlight
Wall tool and **Round Wall** tool located in the Walls tool set	**Wall** tool and **Round Wall** tool, located in the Building Shell tool set, with extended capabilities in the Vectorworks Architect and Landmark products
Modify > Create Polys from Walls	**AEC > Create Polys from Walls** **Landmark > Architectural > Create Polys from Walls** **Spotlight > Architectural > Create Polys from Walls**
Modify > Create Walls from Polygon	**AEC > Create Objects from Shapes** (with extended capabilities) **Landmark > Create Objects from Shapes** **Modify > Create Objects from Shapes** in Spotlight
	AEC > Space Planning > Create Walls from Spaces

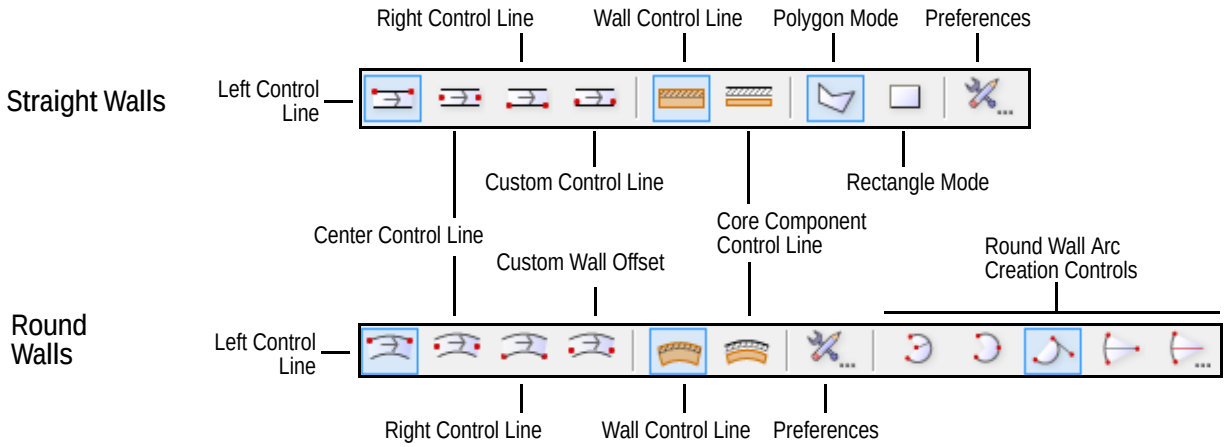
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Walls

Use the wall tools to draw straight or round standard walls or, in the Vectorworks Architect and Landmark products, curtain walls, and to join those walls to other walls. Each wall segment is treated as a separate object. Either automatically join walls together when drawing them, or join them after drawing by using one of the joining options. Other wall options include the ability to add component lines and fills to walls, cap and trim them, and insert symbols (such as doors and windows) into them. You can also add and delete peaks in a wall, if the elevation changes from one end of the wall to the other.

Additional wall parameters are available only in the Architect and Landmark products. These products add the ability to create, save, and use wall styles and to create new wall or component classes during wall creation and editing. In the Vectorworks Architect product, walls and their components also have the ability to have their top and bottom boundaries set to the levels of the story, or the levels of the story above or below the current wall story.

There are four ways to position the wall in relation to the control line; the control line can be set relative to either the wall or an optional core component. Straight walls can be drawn in either **Polygon** mode or **Rectangle** mode. These modes are activated by the buttons on the Tool bar.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Mode	Description
Left Control Line	Walls are drawn along their left sides
Center Control Line	Walls are drawn from the center
Right Control Line	Walls are drawn along their right sides
Custom Control Line / Custom Wall Offset	Walls are drawn from an offset point specified in wall preferences. Setting the offset to be negative offsets the wall to the right of the center line, a positive offset is to the left. Setting a wall offset is most effective when drawing walls containing components.
Wall Control Line	Sets the control line relative to the wall
Core Component Control Line	Sets the control line relative to the wall component that has been designated as the core component in the wall preferences
Polygon Mode	Draws a straight wall or wall system by clicking at each corner to create a custom shape, similar to the **Polygon** tool
Rectangle Mode	Draws a rectangular wall system with two clicks, similar to the **Rectangle** tool's **Corner to Corner** mode
Preferences	Sets the physical parameters of the wall
Arc Creation Controls	Select the arc creation method to use when drawing round walls; for more information on arc creation modes, see "Creating Arcs" on page 293

See "Wall Direction" on page 506 for details on how the starting point and direction in which the wall is drawn determines the wall's interior and exterior sides.

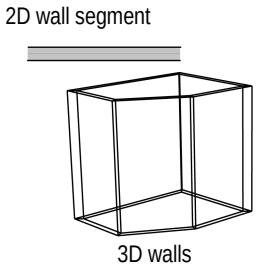
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Drawing Straight Walls
Drawing Round Walls
Wall Direction
Using Wall Styles
Creating Wall End Caps
Creating Wall Components
Creating Walls from Objects
Creating Curtain Walls
Creating Wall Features
Editing Walls
Applying Wall Textures
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Automatically Joining Walls
Framing a Wall
Setting Up the Building Structure with Stories

Drawing Straight Walls

The **Wall** tool creates a hybrid wall object, simultaneously adding both a 2D and a 3D version of the wall to the drawing. Walls can be drawn in Top/Plan or in a 3D view. Standard walls or curtain walls (Vectorworks Architect or Landmark required) can be created.



Create walls by drawing them with the mouse, or by using a mouse-Data bar combination (see "Using the Data Bar" on page 125). The following directions assume that the walls are drawn with the mouse.

If walls are drawn using the Data bar, the **Control Line** mode setting determines whether the dimensions entered are for the left, center, or right edge of the wall, or a custom location.

To create straight walls:

1. Select the **Wall** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set
2. Click the desired **Control Line** mode, and click either **Polygon** or **Rectangle** mode, as desired (see "Creating Walls" on page 497).
3. In the Vectorworks Architect or Landmark product, to draw with an unstyled wall, select <Unstyled> from the Wall Style list on the Tool bar. If the desired styled wall resource has already been created, select it from the Tool bar list or double-click on the resource in the Resource Browser and proceed to Step 6.
4. Click **Preferences** from the Tool bar.

   The Wall Preferences dialog box opens. This dialog box can be accessed any time afterward to modify default wall settings. Walls can also be drawn first, and then the parameters can be set later from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Wall Style (Vectorworks Design Series required)	Select a wall style for the wall; see "Using Wall Styles" on page 507. Alternatively, select <Unstyled> to create the wall without linking the parameters to a wall style. If a selected wall style's parameters are edited in the Wall Preferences dialog box, the wall style automatically changes to unstyled, indicating that the edited style must be saved as a new style to save changes (alternatively, the changes can be applied as an unstyled wall, and they are not saved).

Parameter	Description
Save Preferences as Wall Style (Vectorworks Design Series required)	Saves the current preference settings as a new wall style. The Assign Name dialog box opens; enter a wall style name and click **OK** (see "Creating Wall Styles" on page 508).
Wall Type (Vectorworks Architect or Landmark required)	Select whether to create a standard wall or a curtain wall. Standard walls are described in this section. Curtain walls are described in "Creating Curtain Walls" on page 522.
Preview	Displays a preview of the wall structure, including any defined components; the preview wall is drawn from left to right, so the "top" of the preview, by default, indicates the left part of the wall as it will be drawn. The arrow shows the wall direction.
Overall Thickness	Displays the thickness of the wall. The thickness of a wall with components is defined by the sum of the component thicknesses. When components have been defined, this parameter becomes read-only. If a wall contains no components, enter the wall thickness.
Edit Wall Attributes	Opens the Wall Attributes dialog box, to specify the wall fill, pen, opacity, and wall cap settings
Components	Lists the components that form the structure of the wall, in order from left to right as displayed in the preview. To change the order of a component, click and drag within the # column.
Core	One of the wall components can optionally be designated as a core component by clicking in the Core column. A check mark indicates that the component is the core component. When the **Auto join walls** Vectorworks preference is enabled, components also automatically join based on the core component specified for each wall.
Other Component settings	The remaining columns in the component list display the component thickness and class setting
New	Opens the Wall Component Settings dialog box to define the components of the wall; see "Creating Wall Components" on page 513 for information on creating components
Edit	Opens the Wall Component Settings dialog box to edit the selected component's thickness and attributes (alternatively, double-click on a component to open the Wall Component Settings dialog box)
Duplicate	Duplicates the selected wall component; the duplicate is added to the component list right below the originally selected component
Delete	Deletes the selected wall component; the wall thickness is adjusted accordingly

5. Click **Edit Wall Attributes** to specify the wall attributes.

   The Wall Attributes dialog box opens. The wall attributes of an unstyled wall are initially set to the parameters displayed in the Attributes palette. If they are changed here, the Attributes palette reflects the changed attributes of the selected wall (after exiting the Wall Preferences dialog box).

   Fill, pen, and opacity can be set by class rather than by the attributes in the Wall Attributes dialog box. If the wall class is changed later, the wall changes to use the attributes of the new class. Wall attributes cannot be overridden on a per-instance basis; if a wall style uses class attributes, all walls of that style must use class attributes. However, walls of the same wall style can be placed in different classes.

Click to show/hide the parameters.

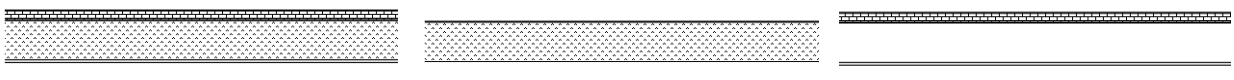
Parameter	Description
Fill	
Style	Specify the wall fill attributes. Select None for no fill, or Class Style to set the wall fill attributes by class rather than by the parameters in the Wall Attributes dialog box. Selected hatch, gradient, or image fills, if not already present in the file, are imported and added to the Resource Browser (default content is automatically imported into the current file at the point of use and displays in the Resource Browser). See “Resource Libraries” on page 219.
Pen	
Style	Specify the wall pen attributes. Select None for no pen, Line Type to select a complex line type, or Class Style to set pen attributes by class rather than by the parameters in the Wall Attributes dialog box.
Thickness	When a pen style has been selected, specify the line thickness. If Line Type is selected as the pen style, also select the line type to use.
Opacity	
Use Class Opacity	Sets wall opacity by class rather than by the parameters in the Wall Attributes dialog box
Opacity	Specifies the transparency of the wall; drag the slider to the left to increase the transparency, or enter a percentage directly in the box to the right of the slider
Wall Caps	
Use Wall Line Attributes	Uses the attributes of the wall line for the wall caps
Use Component Line Attributes	Uses the attributes of the components (left line) for the wall caps

6. Click **OK** to return to the Wall Preferences dialog box. If components are to be added to the wall, click **New** to define each component (see “Creating Wall Components” on page 513). The **Overall Thickness** is then defined by the thickness of the wall components.

   In the Vectorworks Architect and Landmark products, wall components can be assigned to a class; this allows maximum flexibility since the component classes can be shown or hidden separately from the wall class. Select a class for the wall component to control its appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting **New**, or select the class named <Object Class> which places the component in the same class as the wall. By default, components are assigned to the wall class.

   In the Vectorworks Fundamentals product, wall components are automatically assigned to the <Object Class> and are always in the same class as the wall.

   When a component is in an invisible class, its fill and lines are hidden. Invisible components on the interior and exterior of a wall cause the wall’s lines and fill to adjust to the visible components only, making the wall appear thinner than its actual width. This allows walls to show only their structural components, for example. If all components are invisible, the wall displays at its full thickness, without components.



Wall with all components visible

Wall with only structural component visible

Wall with only finish components visible

The Hide Details preference can be used to hide wall components based on scale (Vectorworks Design Series required); see “Hiding Wall Components” on page 517.

7. Click the Insertion Options tab to set the wall insertion options.

   Curtain walls have slightly different parameters; see “Creating Curtain Walls” on page 522.

Click to show/hide the parameters.

Parameter	Description
Height	
Height	Directly sets the desired height of the wall. When the wall height is determined manually by this method, the **Top Bound** property of the wall is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the wall is bound by the layer wall height value, the wall height updates automatically.
Top Bound	Sets the vertical reference that determines the top of the wall. The **Layer Wall Height** value is set by the design layer (see “Setting Design Layer Properties” on page 165). Additional options are available for the Vectorworks Architect product. The top of the wall can be bound by one of the story levels defined for the story or the story above it. By setting the top of the wall to a level type, if the elevation of the associated story changes, the height of the wall changes automatically to match. The default wall styles automatically assign a top bound level type to the wall, assuming that stories will be used in the file. However, this can be changed by selecting a different option.
Top Offset	Sets the offset of the top of the wall from its specified top bound height.
Bottom Bound	Sets the vertical reference that determines the bottom of the wall; Layer Elevation is the only option available unless the Vectorworks Architect product is installed. Additional options are available for the Vectorworks Architect product. The bottom of the wall can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the wall to a level type, if the elevation of the associated story changes, the height of the wall changes automatically to match. The default wall styles automatically assign a bottom bound level type to the wall, assuming that stories will be used in the file. However, this can be changed by selecting a different option.
Bottom Offset	For the bottom of the wall, sets the offset from the layer elevation
Caps	Select whether a wall segment is capped at the start point, the end point, both ends, or has no caps at all
Class	Specifies the default class for the walls
Control Offset	If using the **Custom Control Line** wall mode, enter the offset value for the control line

8. Click the Textures tab to select textures for the wall parts (Renderworks required).

   Textures for curtain walls are controlled by the frame and panel settings on the Definition tab; see “Creating Curtain Walls” on page 522.

Click to show/hide the parameters.

Parameter	Description
Use Component Textures	Uses the textures defined for the wall components to texture the wall This option can also be selected for an existing wall from the Render tab of the Object Info palette
Use Object Textures	Uses the textures defined below for the wall parts (see "Applying a Texture to an Object" on page 1530 for information on applying textures to an object with parts)
Part	Applies the texture selected in **Texture** to the selected parts of the wall
Revert to Overall	If a part was assigned a texture but it should inherit its texture from Overall instead, select the part and click **Revert to Overall**. The part moves back below the divider, and (from Overall) displays as its texture name.
Texture	Applies the selected texture to the selected **Part**. Textures can also be set from the Render tab of the Object Info palette. Textures applied from the Object Info palette override the textures set here.
No Texture	Does not apply a texture to the selected **Part**
Class Texture	Uses the texture defined by the wall's class. The class texture is set from the Other tab in the Edit Classes dialog box. Any walls with that wall style use class textures for that part of the wall (unless overridden). Class Texture can also be chosen for a selected wall in the Render tab of the Object Info palette.
Choose Texture	Choose a texture for the selected **Part** from the default content or the current file's content (see "Resource Libraries" on page 219). Textures set here override the object class textures.

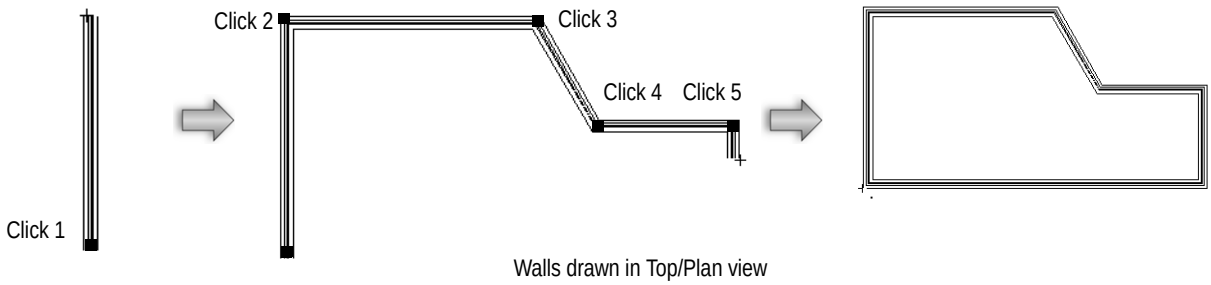
9. Click the Data tab to specify wall record information, which is IFC-compliant and can be included in a wall style schedule (Vectorworks Architect or Landmark required). These fields are optional; enter text only where desired.
10. When the wall parameters have been specified, and any changes saved as a wall style resource if desired (Vectorworks Design Series required), click **OK**.

    In the Design Series, a saved or selected wall style is saved as a resource in the file, and appears in the Resource Browser and in the Wall Style list on the Tool bar. See "Creating Wall Styles" on page 508.
11. If **Polygon** mode is selected, click at the starting point of the first wall section.
12. Click to end the first wall section.

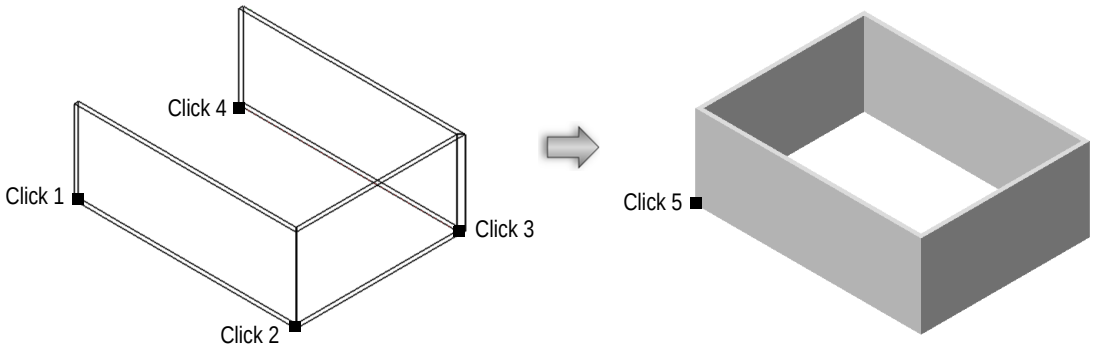
    To continue creating walls, click at the end of each additional wall section.
13. Double-click to finish the wall if the start point and end point are not at the same location; otherwise, click at the starting location (a SmartCursor cue displays) to finish the wall.

    Alternatively, after completing all but the final click, press the keyboard shortcut (K by default) to automatically close the polygonal wall network. See "Modifying Snapping and Mode Shortcuts" on page 1842 to change the shortcut.

Drawing straight walls with **Polygon** mode

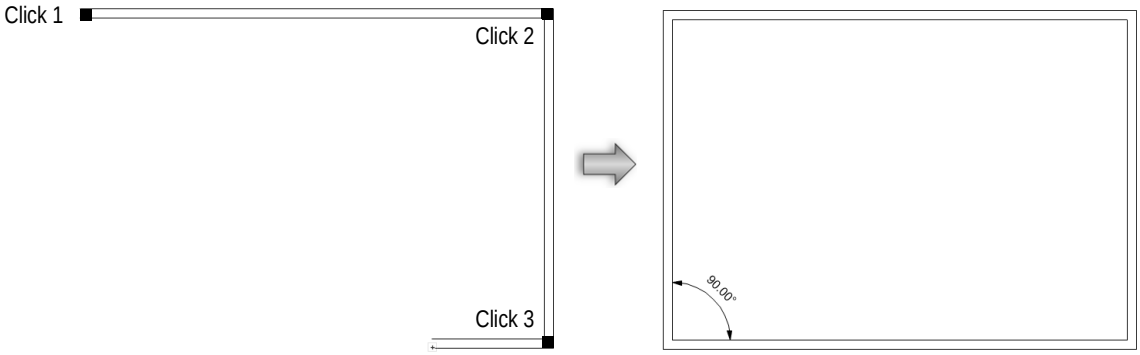


Walls drawn in Top/Plan view



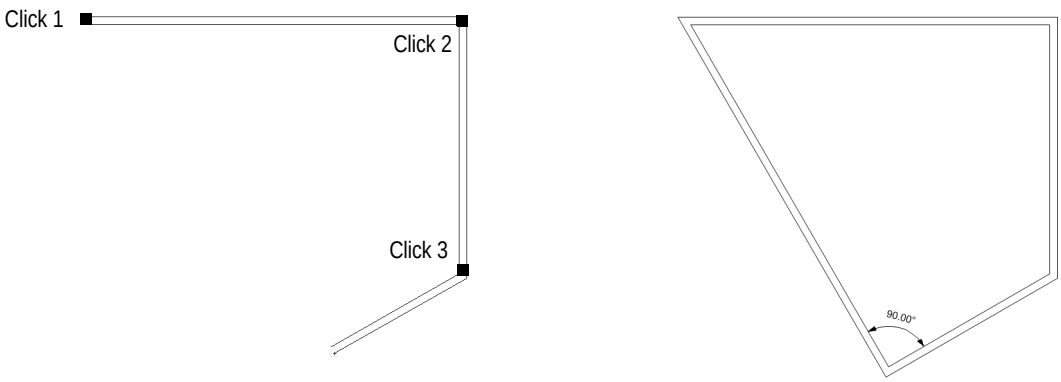
Walls drawn in a 3D view

When drawing a wall network, move the mouse in the direction of the next-to-last click and press the shortcut key before clicking; the software extrapolates the correct alignment and position for a 90° corner and places the final two clicks to complete the wall network.



Move the mouse in the direction of the next-to-last click, and press the shortcut key (K by default)

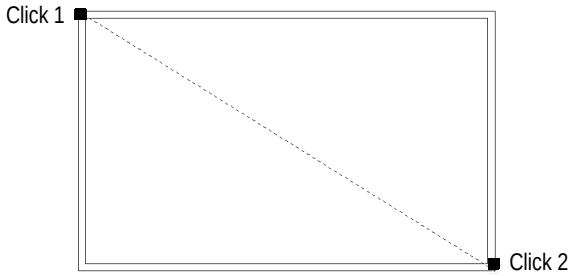
The software places the final two clicks to complete the wall network with a 90° corner



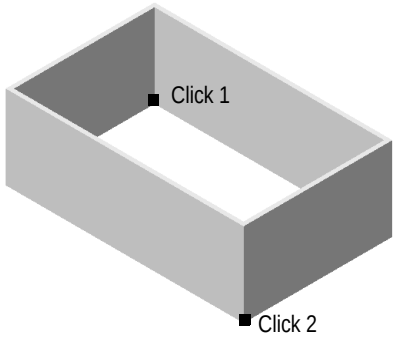
14. If **Rectangle** mode is selected, click at the wall's start point; this becomes one corner of the rectangular wall system. Move the mouse to the opposite corner until the desired size is previewed.
15. Click to set a corner point on the wall system. Four walls are created.

    When the Vectorworks preference **Auto join walls** is on, walls drawn in **Rectangle** mode that overlap or touch each other interact, so complex wall systems can be drawn quickly. See "Automatically Joining Walls in Rectangle Mode" on page 543 for the rules that define these interactions.

Drawing straight walls with Rectangle mode



Walls drawn in Top/Plan view



Walls drawn in a 3D view

The properties of a wall can be edited in the Object Info palette; see "Wall Properties" on page 535.

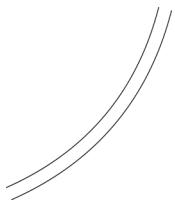
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Walls
Creating Curtain Walls
Wall Direction
Using Wall Styles
Automatically Joining Walls
Creating Wall Components
Editing Walls
Creating Walls from a Polygon
Applying Wall Textures
The Attributes Palette

Drawing Round Walls

Round, 2D-3D hybrid walls can be created and joined to straight walls. The **Round Wall** tool is essentially a combination of the **Wall** tool and **Arc** tool functions. It creates a round wall with the same features and elements as straight walls. Walls can be drawn in Top/Plan or in a 3D view. Standard walls or curtain walls can be created

3D view



2D view

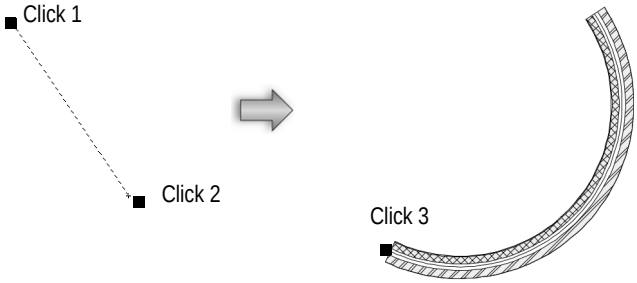
Round walls preferences include all of the same parameters as straight walls.

To create round walls:

1. Select the **Round Wall** tool from the appropriate tool set:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Fundamentals workspace: Walls tool set
- Design Series workspaces: Building Shell tool set

2. Click the desired **Offset** mode (see “Creating Walls” on page 497).
3. Click **Preferences** (parameters are described in “Drawing Straight Walls” on page 499.)
4. Click **OK** when the round wall preferences have been set.
5. Click to set the center point of the wall arc.
6. Click the mouse to begin drawing the wall, or use the Data bar to enter an angle. For more information on arc creation modes, see “Creating Arcs” on page 293.
7. Click to end the wall.



Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

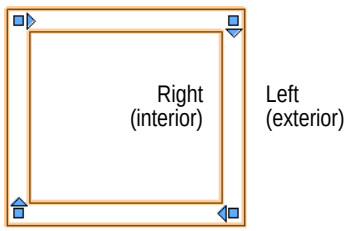
Wall Direction
Automatically Joining Walls
Creating Wall Components
Editing Walls

Wall Direction

The starting point and the direction in which the wall is drawn determine a wall’s “sides.” Think of yourself standing at the wall’s start point and then walking along the top of the wall in the direction of the next point. The side to your left would be designated the left side, and the side to your right the right side. These left and right side designations are used when applying textures to the walls.

Drawing walls in a clockwise direction is always recommended. The wall styles available in the Vectorworks Design Series expect the exterior of the wall to be on the left side.

In Top/Plan view, arrows indicate the current wall direction



For walls drawn in a clockwise direction, the left side is the exterior

New walls drawn with **Rectangle** mode are drawn in a clockwise direction, regardless of click placement. (See “Automatically Joining Walls in Rectangle Mode” on page 543 for a description of wall direction when the Subtract option is used.)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To reverse the wall direction, click **Reverse Sides** from the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Wall Textures

A L Using Wall Styles

Unstyled walls can be used in the Vectorworks Architect and Landmark products. The attributes of an unstyled wall can be changed from the Attributes palette. The use of wall styles, however, facilitates drawing walls by saving the wall preferences settings so that they can be easily selected as the default style when drawing new walls, or so that they can be applied to existing walls. Wall styles are resources that can be imported into other files and shared as office standards.

A wall style contains a significant amount of information:

- Wall composition and structure (components for standard walls, and frames and panels for curtain walls)
- Insertion options (height constraints, caps, classing)
- Wall attributes (fill, pen, line weight, classing, texture resources (Renderworks required), hatch resources)
- Vertical wall constraints (by layer elevation, layer wall height or, for the Vectorworks Architect program, story layer levels)
- Component constraints, for standard walls
- Other non-geometric data (wall style name, thermal data, product data, and so on)

Because a wall style contains so much information, it can be time-consuming to create one from scratch. However, a variety of wall styles are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219).

Depending on how you use wall styles and which product is in use, usually only one type of wall style, simple or constrained, is needed. Wall styles provided by the **Wall** or **Round Wall** tool in a blank file are simple wall styles that use the conventional method of determining the top and bottom wall height according to the layer elevation or layer wall height. However, constrained wall styles are provided in templates named with a BIM prefix; these are available for Vectorworks Architect users from the **Use document template** list in the Create Document dialog box. These wall styles are constrained to the story level settings in those templates, and they provide additional wall detail, accuracy, and flexibility. More information on using wall styles and story levels with BIM is available for drawings set up with metric units and with imperial units.

A selected wall can be converted to an unstyled wall by selecting Convert to Unstyled Wall from the **Style** list in the Object Info palette. The component settings of unstyled walls can be edited by clicking **Components** from the Object Info palette. When a wall becomes unstyled, it loses all its non-geometric data. An unstyled wall’s properties can be converted into a new wall style with a right-click (Windows) or Ctrl-click (Mac) on the wall; select **New Wall Style from Unstyled Wall**.

The insertion options of a styled wall (including wall height, constraints, class, and caps) are properties that can be changed from the Object Info palette without requiring a new style definition.

Unused wall styles can be purged; see “Purging Items from a File” on page 1011.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
Creating Wall Styles
Editing Wall Styles
Applying Wall Styles
Replacing Wall Styles
Creating Walls
Setting Up the Building Structure with Stories

A L Creating Wall Styles

 To create a straight wall style:

 To create a round wall style:

1. Select the **Wall** or **Round Wall** tool from the Building Shell tool set, and then click **Preferences** from the Tool bar.

   The Wall Preferences dialog box opens.

2. To modify an existing wall style, select the **Wall Style** from either the default content or the current file's content. Alternatively, select Unstyled as the wall style and set the parameters.

3. Specify the wall and component parameters as described in "Creating Walls" on page 497, and "Drawing Straight Walls" on page 499.

   The attributes of an unstyled wall can be saved as a wall style. Set the Eyedropper Preferences to pick up all wall attributes and select the **Pick Up Sets Defaults** option. The **Eyedropper** tool can then be used to pick up wall attributes from an unstyled wall and add the settings to the Wall Preferences dialog box, where the settings can be saved as a style.

4. Click **Save Preferences as Wall Style**.

   The Assign Name dialog box opens.

5. Enter a unique name for the wall style and click **OK**.

   If the wall style name already exists, you are prompted to cancel and select a different name, or replace existing walls with the wall style applied with the edited wall style. If you are replacing wall styles, the Wall Replacement dialog box opens; specify the wall alignment properties.

6. The new wall style is saved with the file and is listed under Wall Styles in the Resource Browser as well as the Wall Style list in the Tool bar.

   A new wall style can also be created by clicking **Resources > New Resource > Wall Style** in the Resource Browser. A wall style created in this way is not associated with the current wall preference setting, but can be applied later.

   Worksheets listing the current wall styles and wall areas in the drawing can be added to the drawing from the **VA Create Schedule** command (in the Architect workspace) or the Resource Browser. From the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder that is included with the Vectorworks Architect product (see "Resource Libraries" on page 219). Drag the Wall Area and/or Wall Style Report worksheet to the drawing. The worksheet is populated with information from the objects in the current drawing.

   A wall style resource can be exported; see "Exporting Custom Resources" on page 234. If the wall style is exported to the default Walls~Slabs folder within the user folder, the exported file must be placed within a subfolder, or it won't be included in the list of available wall styles.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
Editing Wall Styles
Applying Wall Styles
Replacing Wall Styles
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Wall Styles

 To edit a straight wall style:

 To edit a round wall style:

1. Select the wall style from the Resource Browser and click **Resources > Edit**.

   The Edit Wall Style dialog box opens.
2. Edit the wall and component parameters as described in “Creating Walls” on page 497, and “Drawing Straight Walls” on page 499.

   If a new wall style **Name** is specified, it replaces the selected wall style name (similar to selecting **Rename** from the Resources menu).
3. Click **OK**.
4. If walls with that style already exist in the drawing, the Wall Replacement dialog box opens.

   The wall style to apply cannot be selected (the edited style is applied). Select the wall alignment properties (see “Replacing Wall Styles” on page 510).
5. Click **OK** to edit the wall style.

   Changes apply to any existing walls in the drawing with the edited wall style, and will be used for any subsequent walls created with that wall style.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Wall Styles
Replacing Wall Styles

Applying Wall Styles

By a variety of methods, wall styles can be selected for a wall before drawing it, or applied to existing walls. Once a wall style has been selected for a wall, the Attributes palette is no longer available for changing wall attributes; attributes are set as part of the style.

To apply a wall style prior to drawing the wall:

1. Select the **Wall** tool or **Round Wall** tool from the Building Shell tool set.
2. Select the desired style from the Wall Style list on the Tool bar. The available wall styles are from either the default content or the current file’s content.

   Alternatively, click **Preferences** from the Tool bar. In the Wall Preferences dialog box, select the **Wall Style**.

To apply a wall style from the Resource Browser prior to drawing the wall:

1. Ensure that no walls are selected.
2. Select a wall style in the Resource Browser, and click **Resources > Apply**.

   Alternatively, double-click on the wall style in the Resource Browser.
3. The **Wall** tool is automatically made active and the selected wall style is applied to the wall as it is drawn. (If a round wall is desired, select the **Round Wall** tool from the Building Shell tool set.)

To apply a wall style to an existing wall from the Object Info palette:

1. Select one or more walls.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. From the Object Info palette, select the wall **Style** from either the default content or the current file's content; see "Resource Libraries" on page 219.

   To apply a different (default) style, select **Replace**.

To apply a wall style to an existing wall from the Resource Browser:

1. Select one or more walls.
2. Select the wall style from the Resource Browser and click **Resources > Apply**. Alternatively, drag the selected wall style from the Resource browser to the wall and click on the wall selection.

   The Wall Replacement dialog box opens.
3. Select the wall alignment properties. The wall style list is disabled (the style selected in the Resource Browser is applied).
4. Click **OK**.

A styled wall can be converted to an unstyled wall.

To remove a wall style:

1. Select one or more walls to un-style.
2. From the Object Info palette, select **Convert to Unstyled Wall**.

   The wall is released from its style; its attributes can be edited from the Attributes palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
Creating Wall Styles
Editing Wall Styles
Replacing Wall Styles

AL Replacing Wall Styles

Wall styles applied to existing walls or curtain walls can be replaced with a different wall style.

> A wall replacement situation may also occur if pasting a styled wall from another file. You can select a replacement wall style and how to align the styles for the pasted walls.

To replace a wall style:

1. Select one or more walls.
2. From the Object Info palette, select Replace from the **Style** list.

   The Wall Replacement dialog box opens. Specify the new wall style, and then select a component (or the wall) from the current and replacement structure lists. Specify the alignment option for each selection; the preview updates, with a red line showing the replacement alignment.

Click to show/hide the parameters.

Parameter	Description
Wall style list	From the list, select the wall style that will replace the current style
Previews	The left preview shows the current wall style, and the right preview shows the replacement wall style selected in the wall style list. Preview walls are drawn from left to right, so the "top" of the preview, by default, indicates the left part of the wall as it will be drawn.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Align lists	On the left, the current wall structure is listed; the right list displays the replacement wall structure. Select one wall component from each list and specify its alignment options. The previews update with red lines to show how the replacement wall will align with the current wall.
Alignment options	Select an option for aligning the current structure to the replacement structure
Replacement options	Select whether to replace the current settings for wall height, class designations, and texture assignments (Renderworks required for textures) with those of the new wall. Texture replacement does not apply to curtain walls.
Curtain Wall Frames and Panels (curtain walls required)	Specify how to handle frames and panels when replacing a curtain wall style
Update Frames and Panels	Updates the existing frames and panels to use the frame and panel settings defined in the new curtain wall style (custom frame spacing and panel size is not changed)
Delete and Recreate Frames and Panels	Clears all existing frames and panels and recreates the curtain wall using the grid definition of the replacement curtain wall style

If walls with different wall styles were selected for replacement, the current wall preview is blank and components are not listed. Wall alignment can only occur for the left, center, or right of the selected wall(s).

3. Click **OK** to replace the wall style of the selected wall(s).

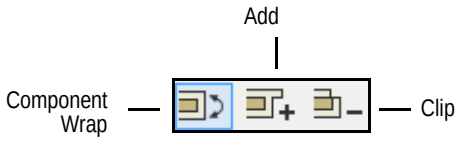
Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Wall Styles
Applying Wall Styles

Creating Wall End Caps

The **Wall End Cap** tool creates both standard and custom component wrapping at the end of a wall. It has three modes:



Mode	Description
Component Wrap	Automatically wraps the selected component, creating a standard wall cap
Add	Creates a custom wall end cap by adding
Clip	Creates a custom wall end cap by removing
Preferences	Sets whether to create wall caps inside or outside the wall endpoint

Setting Wall End Cap Preferences

To set wall end cap preferences:

1. Click the **Wall End Cap** tool from the Building Shell tool set.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Click **Preferences** from the Tool bar.

   The Wall End Cap Settings dialog box opens.

   Determine whether to create the wall end caps outside the wall endpoint. Deselect the option to create the wall end caps inside the wall endpoint, so that the wall length does not extend past the original length.

3. Click **OK**.

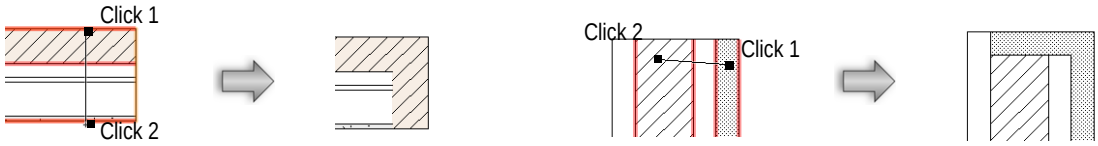
   This option is also available from the Object Info palette for existing wall end caps.

Creating a Standard Wall End Cap

To create a standard end cap by wrapping a component:

1. Click the **Wall End Cap** tool from the Building Shell tool set.
2. Click **Component Wrap** from the Tool bar.
3. Click to select a component within the wall. This component wraps along the wall end.
4. Click to select the component where the wrapping should end.

   The component wraps around the wall end to the end point.



Wall End Cap Properties

An End Cap in Wall object is created by the **Wall End Cap** tool. This object, while distinct from the wall, moves with the wall and adjusts if the wall is resized. The object can be dimensioned, reported in wall schedules, and exported as part of the wall.

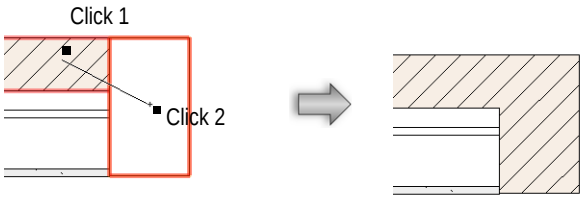
The wall cap uses the line weight of the source component, and its 3D geometry is of the same height as the source component.

The **Extend Cap Past Wall Endpoint** option can be changed from the Object Info palette.

Adding a Custom Wall End Cap

To create a custom wall end cap by adding a shape to a component:

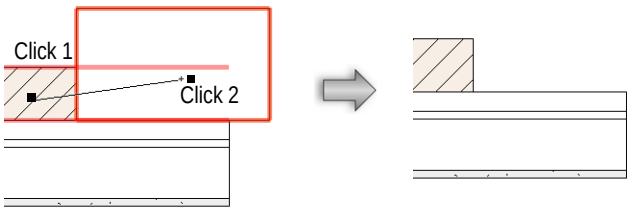
1. Create a 2D, closed shape to represent the wall end cap. The object must intersect the end of the wall.
2. Click the **Wall End Cap** tool from the Building Shell tool set.
3. Click **Add** from the Tool bar.
4. Click to select a component within the wall. This component style is used to create the wall end cap.
5. Click to select the shape.
6. The shape is added to the component and the wall end, creating the custom wall cap.



Clipping to Create a Custom Wall End Cap

To create a custom end cap by clipping a shape from a component:

1. Create a 2D, closed shape to represent the wall end cap. The object must intersect the end of the wall.
2. Click the **Wall End Cap** tool from the Building Shell tool set.
3. Click **Clip** from the Tool bar.
4. Click to select a component within the wall. This component style is used to create the wall end cap.
5. Click to select the shape.
6. The shape is subtracted from the component, creating the custom wall cap.



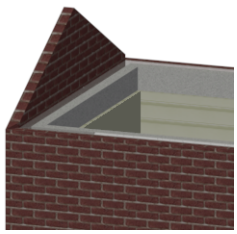
Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Walls
Joining Wall Components
Hiding Wall Components

Creating Wall Components

Wall components define the sections that make up a standard wall. For example, to indicate that a wall is made up of studs, inner drywall, outer sheathing, and then a siding material, define a component for each of these items to illustrate their location. Wall components can be offset at the top or the bottom of the wall, individually constrained relative to the wall, the layer, or, for Vectorworks Architect users, to story levels. Components can be textured (Renderworks required), creating realistic section views and rendered views, as well as accurate wall material estimates. The area and volume of wall components can be calculated in worksheets; see “Worksheet Functions” on page 1344).


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The overall thickness of a wall is equal to the sum of its components. Component fill and pen style are only displayed in Top/Plan view (except for section viewports in the Vectorworks Design Series products).

Use the **Eyedropper** tool to copy wall component settings from one wall to another (see “Transferring Attributes” on page 1095).

Additional wall capabilities are available in the Vectorworks Architect and Landmark products. Curtain walls do not have components.

Wall components can be defined prior to drawing the wall in Wall Preferences mode, or after drawing the wall, from the Object Info palette.

Wall components can also be edited from Wall Preferences mode. Editing a component from wall preferences does not affect existing walls.

Click here for a video tip about this topic (Internet access required).

Defining Components for New Walls

To define a wall component prior to drawing the wall:

1. Select the **Wall** tool or **Round Wall** tool from the appropriate tool set, and then click **Preferences** from the Tool bar.
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set

   The Wall Preferences dialog box opens.
2. On the Definition tab, click **New**.

   The Wall Component Settings dialog box opens. Specify the component thickness, name, and parameters.

Click to show/hide the parameters.

Parameter	Description
Definition	
Name	Provide a name for the component, which displays in the **Components** list in the Wall Preferences dialog box
Class	In the Vectorworks Architect or Landmark product, select a class for the component to control its appearance and visibility (see “Creating Walls” on page 497). The classes present in the drawing are listed; alternatively, create a new class by selecting **New**, or select the class named <Object Class>, which places the component in the same class as the wall object. The Vectorworks Fundamentals product can display classes set in a Vectorworks Design Series product. <Object Class> indicates that the component assumes the same class setting as that of the wall.
Thickness	Specifies the component’s thickness; the thickness of a wall is the sum of its components. A component must have a thickness greater than 0.
Component Top	
Relative to Wall	Sets the top of the component relative to the top of the wall

Parameter	Description
Relative to Story (Vectorworks Architect required to set to story levels)	Select the vertical reference that determines the top of the component from the **Top Bound** list. The Layer Wall Height or Layer Elevation value is set by the design layer (see "Setting Design Layer Properties" on page 165). Additional options are available for the Vectorworks Architect product. The top of the component can be bound by one of the story levels defined for the story or the story above it. Select the level from the **Top Bound** list. By setting the top of the component to a level type, if the elevation of the associated story changes, the height of the component changes automatically to match.
Top Offset	Specifies an additional offset distance for the component from the top of the wall
Follow Top Wall Peaks	The component follows the wall peaks at the top of the wall
Component Bottom	
Relative to Wall	Sets the bottom of the component relative to the bottom of the wall
Relative to Story (Vectorworks Architect required to set to story levels)	Select the vertical reference that determines the bottom of the component from the **Bottom Bound** list. Layer Elevation is the only option available unless the Vectorworks Architect product is installed. Additional options are available for the Vectorworks Architect product. The bottom of the component can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the component to a level type, if the elevation of the associated story changes, the height of the component changes automatically to match.
Bottom Offset	Specifies an additional offset distance for the component from the bottom of the wall
Follow Bottom Wall Peaks	The component follows the wall peaks at the bottom of the wall
Fill	
Style	Specify the wall fill attributes. Select None for no fill, or Class Style to set the component fill attributes by class rather than by the parameters in the Wall Component Settings dialog box. Selected hatch, gradient, or image fills, if not already present in the file, are imported and added to the Resource Browser (default content is automatically imported into the current file at the point of use and displays in the Resource Browser). See "Resource Libraries" on page 219. Use a tile fill with **Fit to Wall** selected, to represent insulation fill. See "Defining Tiles" on page 1112.
Texture (Renderworks required)	
No Texture	Does not apply a texture to the component
Class Texture	Sets the component to use the texture specified by the component's class. The class texture is set from the Other tab in the Edit Classes dialog box; see "Setting Class Properties" on page 179.
Choose Texture	Select a texture for the component from the default content or the current file's content (see "Resource Libraries" on page 219).

Parameter	Description
Left Pen/Right Pen	Select a style for the left and right sides of the component. Select None for no pen, Line Type to select a complex line type, or Class Style to set the component pen attributes by class rather than by the parameters in the Wall Component Settings dialog box.
Thickness	When a pen style has been selected, specify the line thickness; see “Line Thickness Attributes” on page 1101. If Line Type is selected as the pen style, also select the line type to use.

3. Click **OK** to create the component and return to the Wall Preferences dialog box.

   The wall’s **Overall Thickness** value changes to be determined by its components. As components are defined, they display in the preview. Click and drag a component in the # column to change its order.

4. Click **OK**.

Defining Components for Existing Walls

To define or edit wall components for an existing, unstyled wall:

1. Select the wall(s).
2. From the Object Info palette, click **Components**.

   The Components dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Preview	Displays a preview of the wall structure, including the defined components; the preview wall is drawn from left to right, so the “top” of the preview, by default, indicates the left part of the wall as it will be drawn. The arrow shows the wall direction.
Overall Thickness	The thickness of a wall with components is defined by the sum of the component thicknesses
Components	Lists the components that form the structure of the wall, in order from left to right as displayed in the preview. To change the order of a component, click and drag within the # column.
Core	One of the wall components can optionally be designated as a core component by clicking in the Core column. A check mark indicates that the component is the core component. When the **Auto join walls** Vectorworks preference is enabled, components also automatically join based on the core component specified for each wall. See “Automatically Joining Walls” on page 542.
New	Opens the Wall Component Settings dialog box to define the components of the wall as described previously in this section
Edit	Opens the Wall Component Settings dialog box to edit the selected component’s thickness and attributes (you can also double-click on a component to open the Wall Component Settings dialog box)
Duplicate	Duplicates the selected wall component; the duplicate is added to the component list right below the originally selected component
Delete	Deletes the selected wall component; the wall thickness is adjusted accordingly

3. Click **New** or **Edit**, and define or edit the components as described previously.
4. Click **OK** to return to the Components dialog box.

5. Click **OK**. The new component definition is applied to the selected wall(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Wall Textures
The Attributes Palette
Joining Wall Components

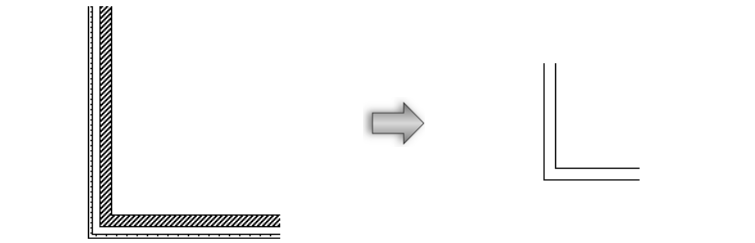
D Hiding Wall Components

The **Hide details** preference prevents wall components from being drawn at or below a pre-set layer scale factor, creating a cleaner drawing when printing at small scales. For example, if the scale factor is set to 1:48, components in walls do not display on any layer set to 1/4” or smaller.

To hide wall components:

1. Select **File > Document Settings > Document Preferences**.

   The Document Preferences dialog box opens.
2. Select **Hide details when layer scale <= 1:**. Enter a scale factor in the scale field.
3. Click **OK**.



Wall components shown at 1:2 scale Wall components hidden at 1:48 scale

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

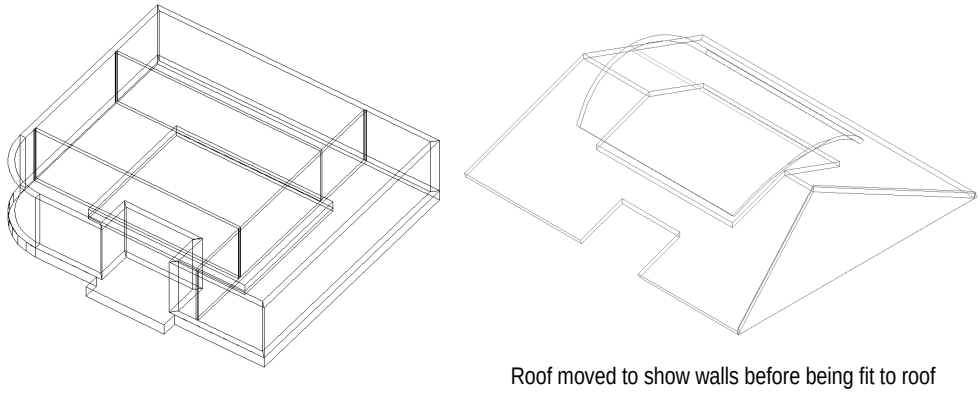
Joining Wall Components
Creating Wall End Caps
Creating Walls

A L Fitting Walls to Defined Geometry

In the Vectorworks Architect and Landmark products, standard walls and curtain walls can be set to the layer height at creation. After creation, they can be automatically extended upward or downward to fit to defining geometry such as the site model, roofs, floors, or NURBS surfaces.

To fit walls to defined geometry:

1. Select the straight or curved wall(s) to be fit.



Roof moved to show walls before being fit to roof

2. With the wall(s) selected, select the **Fit Walls to Objects** command from the appropriate menu.
   - Architect workspace: **AEC > Fit Walls to Objects**
   - Landmark workspace: **Landmark > Architectural > Fit Walls to Objects**

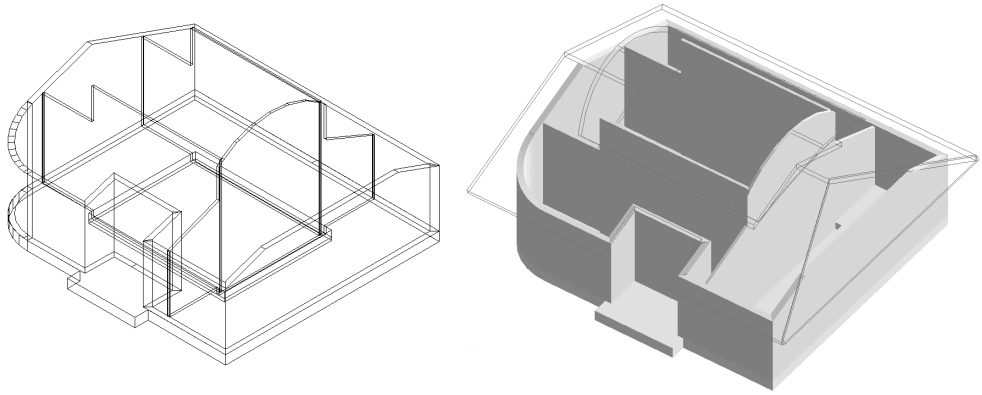
   The Fit Walls to Objects dialog box opens. Specify the location of the wall object and indicate the fit parameters.

Click to show/hide the parameters.

Parameter	Description
Constrain tops of selected walls to objects	Fits walls to objects at the top of the wall(s)
Fit to objects on	Specifies the layer of the defining geometry for the tops of the wall(s); this geometry can consist of roofs, floors, slabs, 3D polygons, extrusions, or NURBS surfaces
Wall top embedding depth	Specifies the distance that the wall penetrates the object. This can avoid problems such as gaps; for very thick walls, a negative embedding depth may be necessary so that the wall does not protrude from the object.
Constrain bottoms of selected walls to objects	Fits walls to object at the bottom of the wall(s)
Fit to objects on	Select the layer of the defining geometry for the bottom of the wall(s)
Include Site Models on selected layer	Select to conform the bottom of the wall(s) to the shape of the underlying site model
Curved wall fit interval	For curved walls, sets the increment for fitting the wall
Wall fitting reference	Select the part of the wall to fit to the constraining object (Center, Left, or Right)

3. Click **OK** to fit the wall(s) to the constraining object indicated.

   Walls or portions of walls which lie outside the constraining object retain their height as set in the Object Info palette. Walls which have had peaks added (with the **Reshape** tool) do not have the peaks reset if the peaks lie outside the constraining object. Walls with a bottom fit to the site model retain a level top rather than a fixed height; the top of the wall does not vary with the bottom.



Creating Walls

Creating Walls from Objects

For flexibility in the design process, walls can be created automatically based on existing objects in the drawing including spaces, polygons and shapes. Likewise, some object types can be created from walls.

A Creating Walls from Spaces

Once the floor plan has been developed with multiple spaces, the interior and exterior walls can be automatically created.

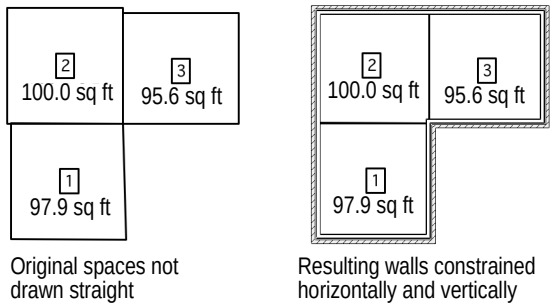
To create walls from spaces:

1. Ensure that space objects are present in the drawing. To automatically create walls with a 3D height, specify a delta-Z value for the design layer where the walls will be created.
2. Select **AEC > Space Planning > Create Walls from Spaces**.

   The Create Walls from Spaces dialog box opens. Specify the style of walls to create and their location.

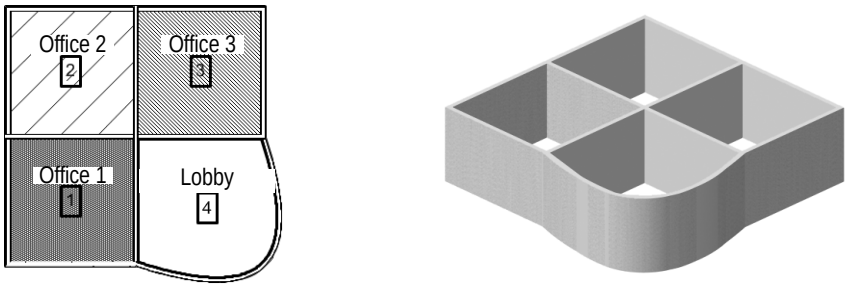
Click to show/hide the parameters.

Parameter	Description
Create Walls from Selected Spaces Only	Select to allow walls to be created from selected spaces; deselect to create walls from all the spaces in the source layer
Source Layer	Specifies the layer containing the space objects
Destination Layer	Indicates the layer on which to create the walls
Place Exterior Walls	Select how to align the exterior walls with the spaces
Along Inside of Spaces	Creates the exterior walls along the inside of the space polylines; select this option if the spaces represent the gross area
Along Outside of Spaces	Creates the exterior walls along the outsides of the space polylines; select this option if the spaces represent the net area
Along Centerline of Spaces	Creates the exterior walls centered along the outside edges of the space polylines
Using Wall Control Offset	Creates exterior walls that are offset as determined by the **Exterior Wall Style**

Parameter	Description
Exterior / Interior Wall Style	Select a wall style for the exterior and interior walls from either the default content or the current file's content
Constrain straight walls to vertical or horizontal	Select to create straight vertical or horizontal walls out of spaces that are not perfectly vertical or horizontal
If they are within ___ degrees of vert/hor	Indicates the threshold for constraining walls; walls that deviate from vertical or horizontal within the number of degrees specified are automatically straightened 
Combine Collinear interior walls	Select to draw multiple, collinear interior walls as a single wall

The available wall styles are defined by wall style resources (see "Using Wall Styles" on page 507).

3. Click **OK** to create the walls on the destination layer.



Exterior and interior walls are applied automatically to the spaces. Round walls approximating the curves are used for Bézier and cubic spline vertices in the space objects.

Resource Libraries
Space Settings
Creating Spaces with the Space Tool
Creating Spaces from Walls

Creating Walls from a Polygon

Walls can be created from a polygon.

In the Vectorworks Design Series products, walls can be created by drawing a polyline or shape and then selecting the **Create Objects from Shapes** command (see "Creating Objects from Shapes" on page 277).

To create walls from a polygon in Vectorworks Fundamentals:

1. Draw or select the polygon to become the basis for the walls.
2. Select **Modify > Create Walls from Polygon**.

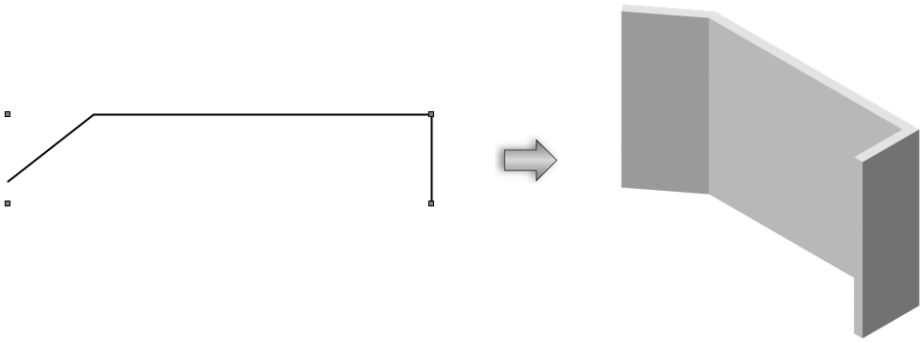
   The Create Walls from Poly dialog box opens. Select the desired wall parameters.

Click to show/hide the parameters.

Parameter	Description
Wall position relative to edge of poly	Specifies whether the wall position should be centered on the polygon, inside the polygon, or outside the polygon
Use existing wall style	Specifies whether to create the walls using the existing wall style
Use existing wall thickness	Specifies whether to create the walls using the existing wall thickness or specify a new **Thickness** value
Use existing wall height	Specifies whether to create the walls using the existing wall height or specify a new **Height** value
Assign to class	Select the class into which the walls should be placed
Delete Source Poly	Deletes the source polygon after the walls are created

3. Click **OK**.

   The wall(s) are created based on the original polygon and the specified parameters.

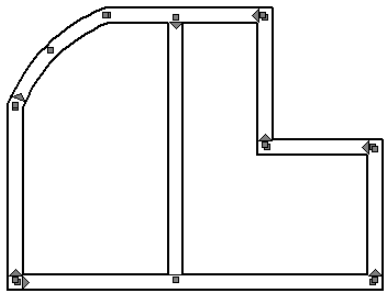


Creating a Polygon or Polyline from Walls

You can create a polygon or polyline based on either the gross area or net area of walls. This is useful for calculating the area of a room, for example, or for using color to differentiate among rooms.

To create a polygon or polyline based on the perimeter of the walls:

1. Select the walls to use for the polygon or polyline. Multiple walls can be selected to create several polygons or polylines at the same time.



2. Select the **Create Polys from Walls** command from the appropriate menu:
   - Fundamentals workspace: **Modify > Create Polys from Walls**
   - Architect workspace: **AEC** > **Create Polys from Walls**
   - Landmark workspace: **Landmark > Architectural > Create Polys from Walls**
   - Spotlight workspace: **Spotlight > Architectural > Create Polys from Walls**

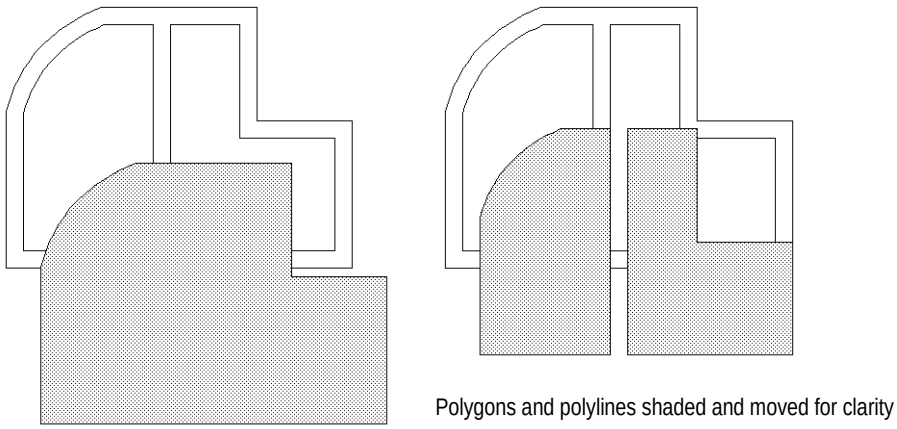
   The Create Polys from Walls dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Gross Area Polys	Creates a polygon or polyline from the exterior perimeter of the selected walls
Net Area (Room) Polys	Creates a polygon or polyline from the interior perimeter of the selected walls

3. Set the parameters and click **OK**.

   The polygon or polyline is created, leaving the original walls unchanged.



Polygons and polylines shaded and moved for clarity

A L Creating Curtain Walls

In the Vectorworks Architect and Landmark products, the capabilities of the **Wall** and **Round Wall** tools are expanded to create curtain walls. Curtain walls consist of frames that contain panels; often, the panels are made of glass, but they can also be opaque or decorative. The Vectorworks program models the frames, panels, and connections according to real-world architectural standards. Since the curtain wall is based on the **Wall** tool, it can take advantage of all the benefits of the **Wall** tool, such as resizing, editing, joining, using walls as the basis of slabs and spaces, bounding by story level, inserting plug-in objects, and more. Doors and windows have special capabilities when they are inserted within curtain walls so that all the design elements work together.

Click here for a video tip about this topic (Internet access required).

To create straight curtain walls:

To create round curtain walls:

1. Select the **Wall** or **Round Wall** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set
2. Click the desired **Offset** mode (see “Creating Walls” on page 497).
3. To draw with an unstyled wall, select <Unstyled> from the Wall Style list on the Tool bar. If the desired styled wall resource has already been created, select it from the Tool bar list or double-click on the resource in the Resource Browser and proceed to Step 6. Curtain walls have specific wall styles; the default styles begin with the prefix CW for easy identification.
4. Click **Preferences** from the Tool bar.

   The Wall Preferences dialog box opens. This dialog box can be accessed any time afterward to modify default wall settings.

   Curtain walls consist of a repeating grid of frames and panels. Specify the number of horizontal and vertical frames that make up one section of the curtain wall, and then specify the appearance of the frames and the panels within that grid. The defined section is repeated over the length and height of the wall.

   Curtain walls can be drawn first, and if not using a wall style, the parameters can be set later from the Object Info palette by clicking **Curtain Wall Grid**. The Curtain Wall Grid dialog box opens, to edit the parameters of the unstyled curtain wall. However, editing unstyled walls by this method may clear existing frames and panels, and recreate the curtain wall grid.

Click to show/hide the parameters.

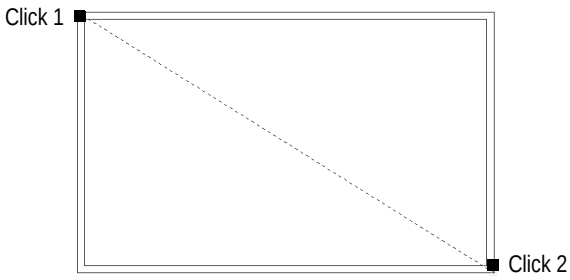
Parameter	Description
Wall Style (Vectorworks Architect or Landmark required)	Select a wall style for the wall; see “Using Wall Styles” on page 507. Alternatively, select <Unstyled> to create the curtain wall without linking the parameters to a wall style. If a selected wall style's parameters are edited in the Wall Preferences dialog box, the wall style automatically changes to unstyled, indicating that the edited style must be saved as a new style to save changes (alternatively, the changes can be applied as an unstyled wall, and they are not saved).
Save Preferences as Wall Style (Vectorworks Architect or Landmark required)	Saves the current preference settings as a new wall style. The Assign Name dialog box opens; enter a wall style name and click **OK** (see “Creating Wall Styles” on page 508).
Wall Type	Select whether to create a standard wall or a curtain wall. Curtain walls are described in this section. Standard walls are described in “Creating Walls” on page 497.
Definition	
Grid Preview	Displays a preview of the grid structure; the preview wall is drawn in a Front view
Vertical/Horizontal Gridlines	In the vertical and horizontal direction, add gridlines to the grid by clicking **Add** and entering the spacing distance. To change the grid order, use the cursor to drag and drop the items in the # column. Remove a selected gridline by clicking **Delete**. The preview area displays the horizontal and vertical gridlines.
Spacing	Sets the distance from the selected gridline to the next gridline; select a gridline to change its value
Add	Adds a horizontal or vertical gridline to the curtain wall frame; by default, the currently selected spacing value is used for the new gridline
Delete	Removes the selected horizontal or vertical gridline from the curtain wall frame
Frame Settings	Opens the Frame Settings dialog box, to set the type of frames and their appearance; see “Setting Curtain Wall Frame Parameters” on page 526
Panel Settings	Opens the Panel Settings dialog box, to set the type of panels and their appearance; see “Setting Curtain Wall Panel Parameters” on page 528
Wall Thickness	Sets the standard frame depth for the curtain wall. This value displays when determining the frame and panel settings, for reference. To avoid errors where walls join, keep the wall thickness and frame thickness values similar.
Offset Reference	Specifies the offset reference point for frame and panel offsets; the actual offset distances are set in the Frame Settings and Panel Settings dialog boxes
Wall Opacity	Sets the opacity of the curtain wall in Top/Plan view
Use Class Opacity	Sets wall opacity by class rather than by the parameters in the Wall Preferences dialog box
Opacity	Specifies the transparency of the wall; drag the slider to the left to increase the transparency, or enter a percentage directly in the box to the right of the slider
Insertion Options	
Height	

Parameter	Description
Height	Directly sets the desired height of the wall. When the wall height is determined manually by this method, the **Top Bound** property of the wall is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the wall is bound by the layer wall height value, the wall height updates automatically.
Top Bound	Sets the vertical reference that determines the top of the wall. The **Layer Wall Height** value is set by the design layer (see "Setting Design Layer Properties" on page 165). Additional options are available for the Vectorworks Architect product. The top of the wall can be bound by one of the story levels defined for the story or the story above it. By setting the top of the wall to a level type, if the elevation of the associated story changes, the height of the wall changes automatically to match. The default curtain wall styles automatically assign a top bound level type to the wall, assuming that stories will be used in the file. However, this can be changed by selecting a different option.
Top Offset	Sets the offset of the top of the wall from its specified top bound height
Bottom Bound	Sets the vertical reference that determines the bottom of the wall; **Layer Elevation** is the only option available unless the Vectorworks Architect product is installed. Additional options are available for the Vectorworks Architect product. The bottom of the wall can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the wall to a level type, if the elevation of the associated story changes, the height of the wall changes automatically to match. The default curtain wall styles automatically assign a bottom bound level type to the wall, assuming that stories will be used in the file. However, this can be changed by selecting a different option.
Bottom Offset	For the bottom of the wall, sets the offset from the layer elevation
Caps	Select whether a wall segment is capped at the start point, the end point, both ends, or has no caps at all
Class	Specifies the default class for the walls
Curtain Wall Cut Plane	Defines the Z-height, relative to the layer where the curtain walls are drawn, for drawing the Top/Plan view of the curtain wall
Control Offset	If using the **Custom Control Line** wall mode, enter the offset value for the control line
Textures	Textures for curtain walls are controlled by the frame and panel settings
Data	Specifies wall record information, which is IFC-compliant and can be included in a wall style schedule. These fields are optional; enter text only where desired.

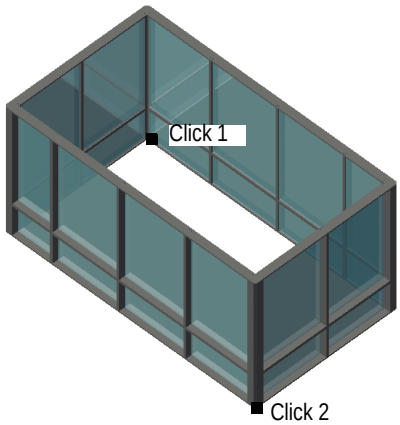
5. When the curtain wall parameters have been specified, and any changes saved as a wall style resource if desired, click **OK**. A saved or selected wall style is saved as a resource in the file, and appears in the Resource Browser and in the Wall Style list on the Tool bar.
6. If **Polygon** mode is selected, click at the starting point of the first curtain wall section.
7. Click to end the first wall section.

   To continue creating curtain walls, click at the end of each additional wall section.

8. Double-click to finish the curtain wall if the start point and end point are not at the same location; otherwise, click at the starting location (a SmartCursor cue displays) to finish the wall.
9. Alternatively, if **Rectangle** mode is selected, click at the wall's start point; this becomes one corner of the rectangular wall system. Move the mouse to the opposite corner until the desired size is previewed.
10. Click to set a corner point on the wall system. Four walls are created.



Drawing straight curtain walls with **Rectangle** mode in Top/Plan view



Setting Curtain Wall Frame Parameters
Setting Curtain Wall Panel Parameters
Editing Curtain Walls
Wall Properties
Creating Walls

Setting Curtain Wall Frame Parameters

The frame settings and appearance are controlled individually for each frame location, for maximum flexibility when setting the frame parameters.

> Curtain walls can be drawn first, and if not using a wall style, the parameters can be set later from the Object Info palette by clicking **Curtain Wall Grid**. The Curtain Wall Grid dialog box opens, to edit the parameters of the unstyled curtain wall. However, editing unstyled walls by this method may clear existing frames and panels, and recreate the curtain wall grid.

To set the curtain wall frame appearance:

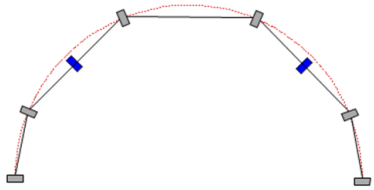
1. From the Definition tab of the Wall Preferences dialog box, or when editing a curtain wall style, click **Frame Settings**.

   The Frame Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Frame category	Select a category of frame element to define, and then set its parameters. The starting and ending frames, top and bottom frames, and the inner horizontal and vertical frames can each be defined separately, for maximum flexibility. Multiple categories can be selected at one time for quickly defining common parameters.
Definition	

Parameter	Description
Type	Select the type of frame • Full-Depth: Standard, normal frame • Capped Full-Depth: Standard frames with caps; specify the **Cap Depth** • Butt-Glazed: Standard frame that is concealed behind the panels, creating the appearance that the panels are joined without a frame • Brake Metal: Corner frame, or segmented frame for round walls; in a straight wall, behaves like a full-depth frame • Capped Brake Metal: Combines the properties of a capped frame and brake metal frame; specify the **Cap Width** if different from the overall width • Virtual: Hidden frame; this is useful for piecing together curtain wall units, especially when vertically stacked. A virtual top frame appears to butt up against the bottom frame of the curtain wall above it.
Preview	Displays a labeled diagram of the selected frame type; the numbers indicated on the diagram correspond to the labeled parameters below the preview, for easy identification
Width	Sets the width of the frame
Depth	Select how the frame depth is specified. If set to **Use Wall Thickness**, the frame is set to the thickness of the curtain wall as specified in the Curtain Wall Grid or Wall Preferences dialog box (the thickness is displayed). Alternatively, set a **Custom Depth** and enter the value.
Panel Inset	Determines the overlap between the frame and the neighboring panels, for both Top/Plan and 3D views. The allowable inset distance can vary from 0, for a frame without a notch, to half the width of the frame, where the panels meet in the middle of the frame.
Cap Depth (capped types only)	Specifies the cap depth, relative to the frame depth
Cap Width (capped brake metal type only)	Sets the width of the cap for a brake metal frame, allowing the cap and the frame to have different widths
Offset Reference	Displays the offset reference that was selected in the Curtain Wall Grid or Wall Preferences dialog box
Offset	Shifts the frame position relative to the offset reference. Enter a positive value to shift the frame to the right side of the wall as drawn, or enter a negative value to shift the frame to the left side of the wall. To center the frame on the wall, select Center Line as the **Offset Reference** in the Curtain Wall Grid or Wall Preferences dialog box, and use an offset of 0.
Half Width on Wall Boundary	Creates a frame of half the width on the wall boundary. This is useful for piecing together curtain wall units, especially for end-to-end curtain wall joins. This option only applies to capped full depth, full depth, and butt-glazed frames that lie on the start, end, top, or bottom wall boundary.

Parameter	Description
Segment Round Wall (round wall required)	This parameter only applies to round curtain walls with full-height, vertical frames. Normally, this option should be selected, to create a wall with straight panels that follow the arc shape, for a rounded effect. However, by deselecting this option for certain standard or corner frames within a curtain wall, a straight segment can be created that does not follow the arc, for a straight panel of glass that is part of the rounded wall.  The two blue frames have the Segment Round Wall option deselected, resulting in a straight section of panels
Frame/Cap Class	The frame and the caps in each frame category can be placed in distinct classes to control their appearance and visibility. Select the class to use from the current list of classes, create a new class, or select <Object Class> to place the frame or cap in the same class as the curtain wall.
Frame/Cap Attributes	Each attribute can be set by the class style or to specific settings. If set by the class style, the attributes are controlled by the class selected for the curtain wall, frame, or cap.
Frame/ Cap Fill	Select Class Style to use the class style for the fill attribute, choose None for no fill, or choose a fill style
Frame/Cap Pen	Select Class Style to use the class style for the pen attribute, choose None for no pen, or choose a solid/dash or pattern style
Frame/Cap Texture (Renderworks required)	Click No Texture to apply no texture, Class Texture to use the class texture for the frame or cap, or click Choose Texture to select the texture to apply from either the default content or the current file's content; see "Resource Libraries" on page 219
IFC	Attaches IFC data to the curtain wall frames for exporting to IFC format

2. Click **OK**.

   The parameters can be edited from the Object Info palette; see "Wall Properties" on page 535. The **Cut Plane** parameter determines the height for displaying the curtain wall in Top/Plan view.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Curtain Walls
Editing Curtain Walls
Applying Object Attributes
Assigning IFC Data to Objects

AL Setting Curtain Wall Panel Parameters

The panel settings and appearance are specified in the Panel Settings dialog box.

If setting up curtain wall preferences, these parameters are available from the Definition tab of the Wall Preferences dialog box. For an existing curtain wall, they are available from the Curtain Wall Grid dialog box.

To set the curtain wall panel appearance:

1. From the Object Info palette of a selected curtain wall, click **Curtain Wall Grid**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Curtain Wall Grid dialog box opens.

2. Click **Panel Settings**.

   The Panel Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Definition	
Type	Select the kind of panel to place between the frames. Glazed panels are typical glass panels, and spandrel panels are opaque. Spandrel face panels are decorative panels. Open panels provide empty space between the frames.
Preview	Displays a labeled diagram of the selected panel type; the numbers indicated on the diagram correspond to the labeled parameters below the preview, for easy identification
Thickness	Specifies the panel thickness. The inset is controlled by the **Panel Inset** parameter in the Frame Settings dialog box. The combination of panel **Thickness** and **Panel Inset** parameters creates the channel for the panel inserted in the frame.
Use Infill (spandrel and spandrel face only)	Allows the area to the right (inside of the curtain wall as drawn) and between the frames to be filled. The attributes of the infill panel are determined by the panel.
Infill Depth	Specifies the additional depth of the infill panel
Face Depth (spandrel face only)	Sets the depth of the spandrel face
Wall Thickness	Displays the **Wall Thickness** value from the Curtain Wall Grid or Wall Preferences dialog box for reference
Offset Reference	Displays the offset reference that was selected in the Curtain Wall Grid or Wall Preferences dialog box
Offset	Shifts the panel position relative to the offset reference. Enter a positive value to shift the panel to the left side of the wall as drawn, or enter a negative value to shift the frame to the right side of the wall. To center the panel on the wall, select Center Line as the **Offset Reference** in the Curtain Wall Grid or Wall Preferences dialog box, and use an offset of 0.
Panel/Face Class	The panel and the spandrel face (if selected as a panel type) can be placed in distinct classes to control their appearance and visibility. Select the class to use from the current list of classes, create a new class, or select <Object Class> to place the panel or spandrel face in the same class as the curtain wall.
Panel/Face Fill	Select Class Style to use the class style for the fill attribute, choose None for no fill, or choose a fill style
Panel/Face Pen	Select Class Style to use the class style for the pen attribute, choose None for no pen, or choose a solid/dash or pattern style
Panel/Face Texture (Renderworks required)	Click No Texture to apply no texture, Class Texture to use the class texture for the panel or spandrel face, or click Choose Texture to select the texture to apply from either the default content or the current file's content; see "Resource Libraries" on page 219
IFC	Attaches IFC data to the curtain wall panels for exporting to IFC format

3. Click **OK**.

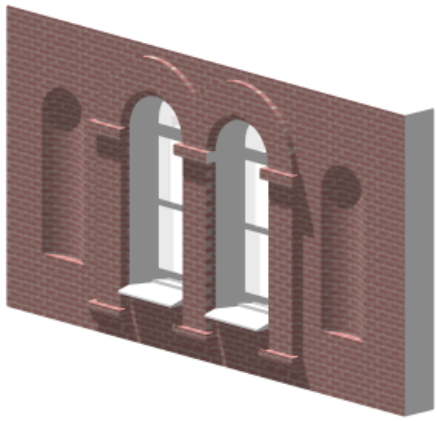
Creating Curtain Walls
Editing Curtain Walls
Applying Object Attributes
Assigning IFC Data to Objects

Creating Wall Features

Once a standard round or straight wall has been created, features such as projections and recesses can be added to it. Create wall features from 2D or 3D objects, as well as symbols. By default, 2D objects are extruded to span the height of the wall; offsets from the top and bottom of the wall can be specified as needed. Use 3D objects, such as sweeps or other solids, to create more complicated features. Use symbols to be able to update multiple instances of a feature with a single edit.

> It it not recommended to create wall features that are applied to the entire length of the wall, such as battered walls, or wall caps.

Unlike a symbol, a wall feature is actually part of the wall geometry. A wall feature interacts with the wall components, it renders along with the rest of the wall, and if it was created from a 2D object, it automatically adjusts when the wall height changes. Because wall features interact with wall components, wall features cannot be created for curtain walls.



Click here for a video tip about this topic (Internet access required).

Creating Wall Projections
Creating Wall Recesses
Wall Feature Properties
Editing Wall Features
Creating Walls

Creating Wall Projections

To create wall projections:

1. Create the closed 2D or 3D solid object with which to modify the wall. The geometry of the object must define an area or volume that has no self-contacts, no self-intersections, and no edges that fold back on themselves.
2. For a 2D object, simply make sure that the object overlaps the wall, and that the view is Top/Plan. For a 3D object, first create the object at the appropriate height in relation to the bottom of the wall (Z axis); then set the appropriate X and Y axis location so that the object overlaps the wall.

3. With both the wall and the modifier object selected, choose the **Create Wall Projection** command from the appropriate menu:
   - Architect workspace: **AEC > Create Wall Projection**
   - Landmark workspace: **Landmark > Architectural > Create Wall Projection**
   - Spotlight workspace: **Spotlight > Architectural > Create Wall Projection**

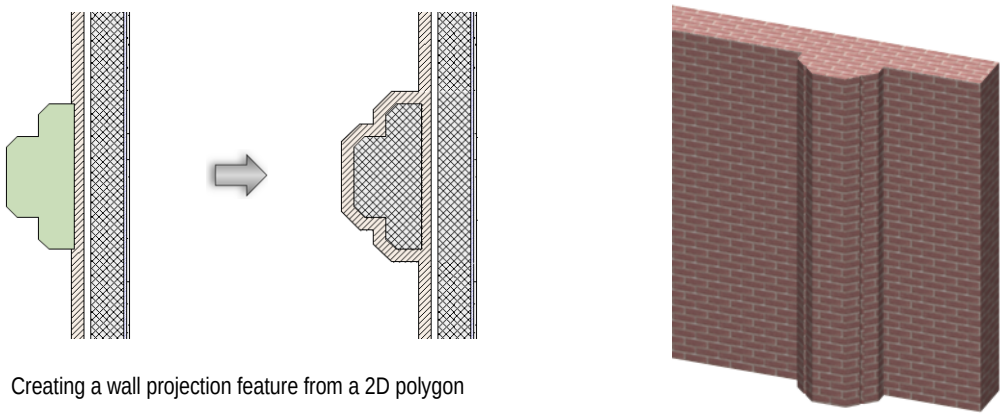
   Alternatively, for 2D objects only, select **Modify > Add Surface**.

   The Create Wall Feature Projection dialog box opens; different fields display depending on whether a 2D or 3D modifier object was selected.

Click to show/hide the parameters.

Parameter	Description
Feature Component	From the list of components currently in the wall, select a component to be applied to the area of the feature object. Otherwise, select <Object> to use the object's own fill, or select <None> if the feature is intended to be a hole in the wall (for example, to create an opening for an object that is not part of the wall).
Left Side Wrap (2D objects only)	Select the first component to wrap around the feature on the left side of the wall; this can be <None>, or any component to the left of the **Feature Component** (higher in the list). All components to the outside of the **Left Side Wrap** component also wrap.
Right Side Wrap (2D objects only)	Select the first component to wrap around the feature on the right side of the wall; this can be <None>, or any component to the right of the **Feature Component** (lower in the list). All components to the outside of the **Right Side Wrap** component also wrap.
Offset From Wall Top (2D objects only)	Specifies the distance from the top of the wall where the wall feature ends in 3D views
Use Profile as Core (2D objects only)	Select this option to use the area of the original 2D object as the core of the wall feature; other wall components will wrap around the core, which may make the projection larger than the original object. Deselect this option to limit the area of the projection to the area of the original 2D object.
Cut Plane (3D objects only)	Specifies the distance from the bottom of the wall at which a section slice is taken of the 3D object and is used to represent the wall feature in Top/Plan view. If the 3D object does not run the whole height of the wall, be sure that this value actually intersects the 3D object; otherwise the object will not appear in Top/Plan view.
Use Object 3D Attributes (3D objects only)	Select this option to display the surfaces created by the feature object with its own attributes when it is rendered in 3D; when deselected, the wall's attributes are used on those surfaces
Show Extents Below/ Above Cut Plane (3D objects only)	In addition to the 3D section specified by the **Cut Plane** value, shows the outline of the part of the 3D object below and above the cut plane; the pen attributes of this representation can be edited (see "Wall Feature Properties" on page 533)
Preview	Changes the view of the wall feature to match the current parameter settings, without saving the settings

4. Set the parameters and click **OK**.



Creating a wall projection feature from a 2D polygon

Creating Wall Recesses
Wall Feature Properties
Editing Wall Features
Creating Wall Features
Creating Walls

Creating Wall Recesses

To create a wall recess:

1. Create the closed 2D or 3D solid object with which to modify the wall. The geometry of the object must define an area or volume that has no self-contacts, no self-intersections, and no edges that fold back on themselves.
2. For a 2D object, simply make sure that the object overlaps the wall in Top/Plan view. For a 3D object, first create the object at the appropriate height in relation to the bottom of the wall (Z axis); then set the appropriate X and Y axis location so that the object overlaps the wall.
3. With both the appropriate wall and the modifier object selected, choose the **Create Wall Recess** command from the appropriate menu:
   - Architect workspace: **AEC > Create Wall Recess**
   - Landmark workspace: **Landmark > Architectural > Create Wall Recess**
   - Spotlight workspace: **Spotlight > Architectural > Create Wall Recess**

   Alternatively, for 2D objects only, select **Modify > Clip Surface**.

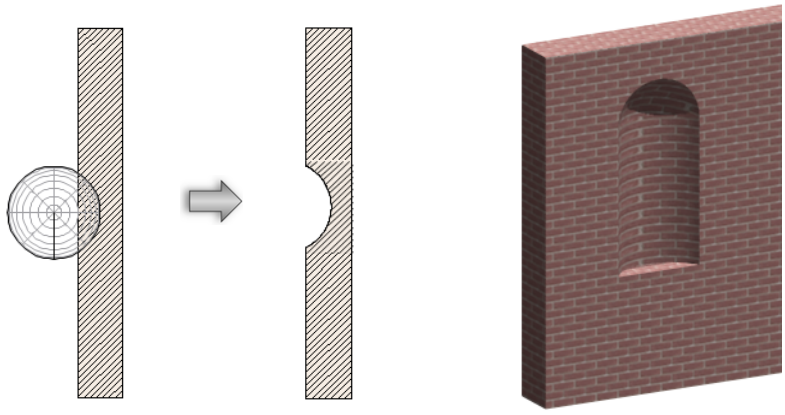
   The Create Wall Feature Recess dialog box opens; different fields display depending on whether a 2D or 3D modifier object was selected.

Click to show/hide the parameters.

Parameter	Description
Left Side Wrap (2D objects only)	Select the first component to wrap around the feature on the left side of the wall; this can be <None>, or any component to the left of the **Feature Component** (higher in the list). All components to the outside of the **Left Side Wrap** component also wrap.
Right Side Wrap (2D objects only)	Select the first component to wrap around the feature on the right side of the wall; this can be <None>, or any component to the right of the **Feature Component** (lower in the list). All components to the outside of the **Right Side Wrap** component also wrap.

Parameter	Description
Offset From Wall Top (2D objects only)	Specifies the distance from the top of the wall where the wall feature ends in 3D views
Cut Plane (3D objects only)	Specifies the distance from the bottom of the wall at which a section slice is taken of the 3D object and is used to represent the wall feature in Top/Plan view. If the 3D object does not run the whole height of the wall, be sure that this value actually intersects the 3D object; otherwise the object will not appear in Top/Plan view.
Preview	Changes the view of the wall feature to match the current parameter settings, without saving the settings

4. Select the options for the wall feature, and click **OK**.



Creating a wall recess feature from a 3D sweep

Creating Wall Projections
Wall Feature Properties
Editing Wall Features
Creating Wall Features
Creating Walls

Wall Feature Properties

After a wall feature has been created, it can be edited in the Object Info palette. Alternatively, right-click (Windows) or Ctrl-click (Mac) on the feature and select **Properties** from the context menu.

In addition to the parameters available when the feature is created, the following options are available.

Click to show/hide the parameters.

Parameter	Description
Flip	Flips the wall feature to the other side of the wall
Set Position	Activates the **Move by Points** tool in Reference Point mode, to move the wall feature a specified distance along the length of the wall (see “Moving Objects with the Reference Point Mode” on page 1008)

Parameter	Description
Edit Extents Attributes (3D object projections only)	If **Show Extents Below Cut Plane** or **Show Extents Above Cut Plane** is selected, opens the Extents Attributes dialog box to allow editing of the 3D object outlines shown for the areas below and above the cut plane

Creating Wall Projections
Creating Wall Recesses
Editing Wall Features
Creating Wall Features
Creating Walls

Editing Wall Features

Similar to symbols, the original 2D or 3D object (or symbol) can be edited after the wall feature is created. For example, the location or shape of the object can be changed.

To access the editing mode, double-click the wall feature. Alternatively, right-click (Windows) or Ctrl-click (Mac) on the feature and select **Edit** from the context menu. To delete the wall feature, delete the original object from which the wall feature was created. Click **Exit Wall Feature** to exit the editing mode.

Also similar to symbols, wall features can be moved and duplicated in various ways in both straight and round walls. See the following topics:

- “Moving Symbols in Walls with the Selection Tool” on page 256
- “Nudging Symbols in Walls” on page 257
- “Moving Symbols in Walls with the Move Command” on page 257
- “Moving Symbols in Walls with the Move by Points Tool” on page 258
- “Duplicate Array” on page 1014

Creating Wall Projections
Creating Wall Features
Creating Walls

Editing Walls

Wall Properties

Edit the properties of selected straight or round walls and their components, as well as curtain walls (Vectorworks Architect or Landmark required), in the Object Info palette. To change the wall attributes, use the Attributes palette. Certain parameters are available for standard walls or for curtain walls only. Settings on the Render tab (Renderworks required) are not available for curtain walls, since the textures of frames and panels are specified separately. Curtain walls use the **Edit Curtain Wall** tool to adjust the frames and panels.

Click to show/hide the parameters.

Parameter	Description
Shape tab	
Style	In the Vectorworks Fundamentals product, all walls are unstyled. The Vectorworks Architect or Landmark product is required to select other wall styles; see “Creating Walls” on page 497.
Thickness	
Thickness	Displays the overall thickness of the wall; the thickness of a wall without components is determined by the wall attributes. The thickness of a wall with components is defined by the sum of the component thicknesses.
Vis Thick	Displays the visible wall thickness, while accounting for component visibility settings. Invisible components on the interior and exterior of the wall cause the wall to appear thinner than its actual length, for display purposes.
Height	
Height	Directly sets the desired height of the wall. When the wall height is determined manually by this method, the **Top Bound** property of the wall is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the wall is bound by the layer wall height value, the wall height displays automatically.
Top Bound	Sets the vertical reference that determines the top of the wall. The **Layer Wall Height** value is set by the design layer (see “Setting Design Layer Properties” on page 165). Additional options are available for the Vectorworks Architect product; see “Creating Walls” on page 497.
Top Offset	Sets the offset of the top of the wall from its specified **Top Bound** height.
Bottom Bound	Sets the vertical reference that determines the bottom of the wall; **Layer Elevation** is the only option available unless the Vectorworks Architect product is installed
Bottom Offset	For the bottom of the wall, sets the offset from the layer elevation
Caps	Select whether a wall segment is capped at the start point, the end point, both ends, or has no caps at all
Attr (standard wall required)	When the wall is capped, specifies whether the wall cap attributes comes from the wall line attributes or the component line attributes
Components (standard wall required)	Edits the wall components of unstyled walls (see “Creating Wall Components” on page 513)

Parameter	Description
Cut Plane (curtain wall required)	Sets the height for the display of the Top/Plan appearance of the curtain wall
Curtain Wall Grid (curtain wall required)	Opens the Curtain Wall Grid dialog box, for editing the curtain wall frame and panel settings as described in “Creating Curtain Walls” on page 522
Reverse Sides	Reverses the direction of the wall sections (see “Wall Direction” on page 506); walls should be drawn in a clockwise direction
Render tab (Renderworks, standard walls required)	
Mode	Select whether to texture the wall according to textures defined for the wall (By Object) or according to textures defined for the components (By Component). See “Applying a Texture to an Object” on page 1530, “Applying Wall Textures” on page 1544, and “Creating Wall Components” on page 513.

Reshaping Walls
Removing Wall Breaks
Joining Walls
Inserting and Editing Symbols Within a Wall
Editing Curtain Walls
Creating Walls
Creating Curtain Walls

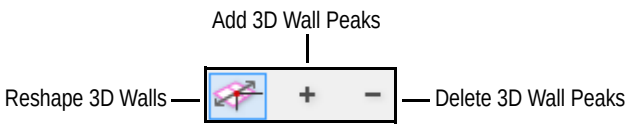
Reshaping Walls

Use the **Reshape** tool in a view other than Top/Plan to edit the wall height, change the elevation of wall peaks, add vertices to create peaks in a wall, and delete vertices that have been added. Use the **Selection** tool to change the wall length. Symbols remain where placed when a wall is reshaped.

In the following situations, the **Reshape** tool displays 2D Reshape modes; the vertices at either end of the wall can be moved, to change the wall length and/or location. See “2D Reshape Modes” on page 1044.

- In Top/Plan view
- When multiple walls are selected
- When a wall and other objects are selected
- When a marquee has been drawn

In any 3D view, when only a single wall is selected, the **Reshape** tool has three modes available.



Mode	Description
Reshape 3D Walls	Adjusts the position of a selected wall vertex
Add 3D Wall Peaks	Adds a vertex to a wall for reshaping purposes
Delete 3D Wall Peaks	Deletes a wall vertex

Walls can also be trimmed using the Trim tool; see “Trim Tool” on page 1060.

Changing Wall Length
Changing Wall Height
Changing Wall Peak Height
Adding a Vertex to Walls
Deleting a Wall Vertex
Reshaping Objects
Changing Round Wall Radius
Removing Wall Breaks
Joining Walls
Joining Walls with the Fillet Tool
Inserting and Editing Symbols Within a Wall
Creating Walls

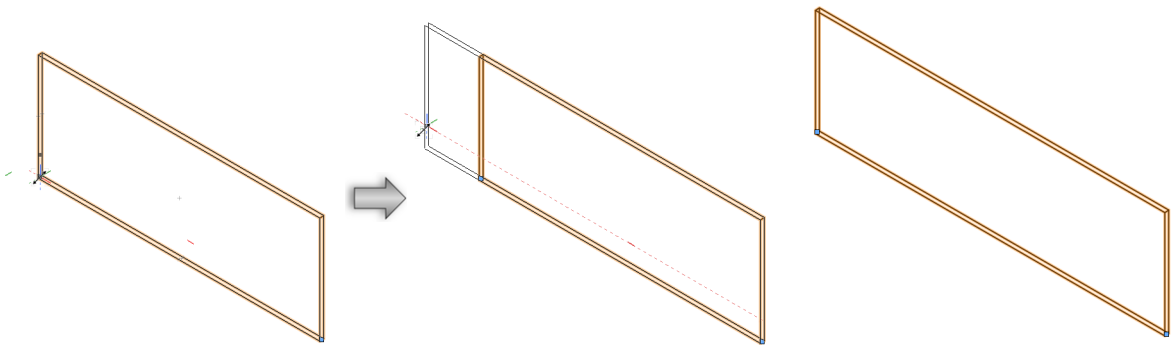
Changing Wall Length

To change a wall's length:

1. Select the wall to reshape.
2. Click the **Selection** tool from the Basic palette.

   Single Object Interactive Scaling mode must be enabled.
3. Position the cursor over an end selection handle, and click.

   In a 3D view, the handles are at the base of the wall.
4. Move the mouse to lengthen or shorten the wall.
5. Click when the wall is at the desired length.



With the Selection tool, drag the handle at the base of the wall to lengthen or shorten the wall

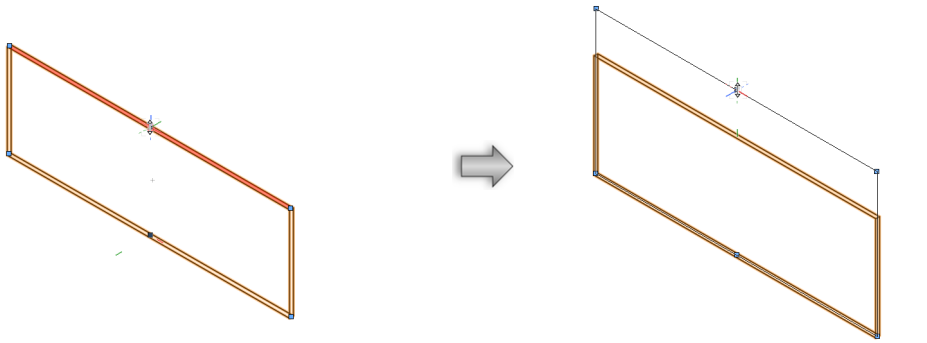
Wall length can also be changed with the **Reshape** tool in Top/Plan view. In a 3D view, draw a marquee with the **Reshape** tool to enclose the wall vertex in a marquee that is co-planar with the bottom wall elevation. Select the wall with the **Reshape** tool. The available modes switch to the 2D reshape modes, and 2D reshape functionality is enabled (see "2D Reshape Modes" on page 1044). This allows the wall length to be adjusted, even when in a 3D view.

Reshaping Walls
The Selection Tool
Reshaping Objects
Trim Tool

Changing Wall Height

To change the height of the wall:

1. In a 3D view, select the wall to reshape.
2. Click the **Reshape** tool from the Basic palette, and select **Reshape 3D Walls** mode.
3. Position the cursor over one of the handles in the middle of the top or bottom of the wall, and click.
   When the cursor is over a reshape handle, the standard arrow cursor changes into a double-headed, unfilled arrow.
4. Move the mouse to adjust the wall.
5. Click at the desired location.



With the 3D Reshape cursor, drag a handle in the middle to change the wall height

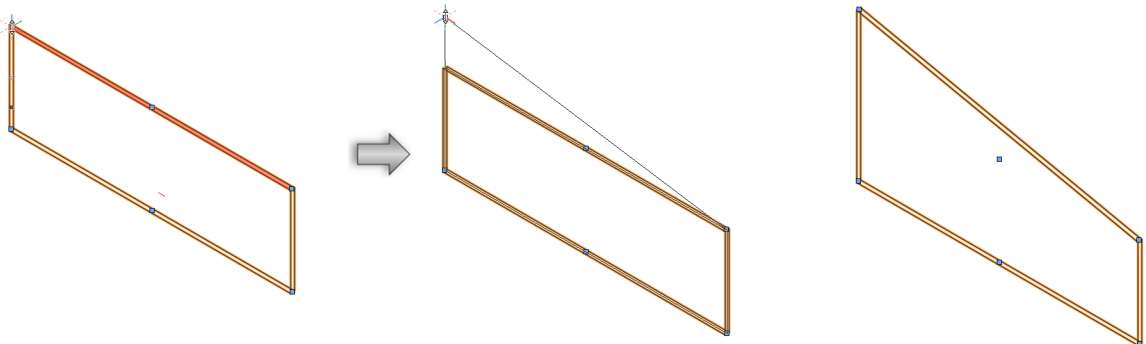
In the Object Info palette, the height change displays as a top or bottom offset value.

Reshaping Walls

Changing Wall Peak Height

To change the height of a peak on the wall:

1. In a 3D view, select the wall to reshape.
2. Click the **Reshape** tool from the Basic palette, and select **Reshape 3D Walls** mode.
3. Position the cursor over one of the handles on the ends of the top or bottom of the wall, and click.
   When the cursor is over a reshape handle, the standard arrow cursor changes into a double-headed, unfilled arrow.
4. Move the mouse to adjust the wall.
5. Click at the desired location.



With the 3D Reshape cursor, drag a handle on the end of the top or bottom of the wall to reshape the wall

Changing the height of a wall peak does not change the height of the wall.

Reshaping Walls

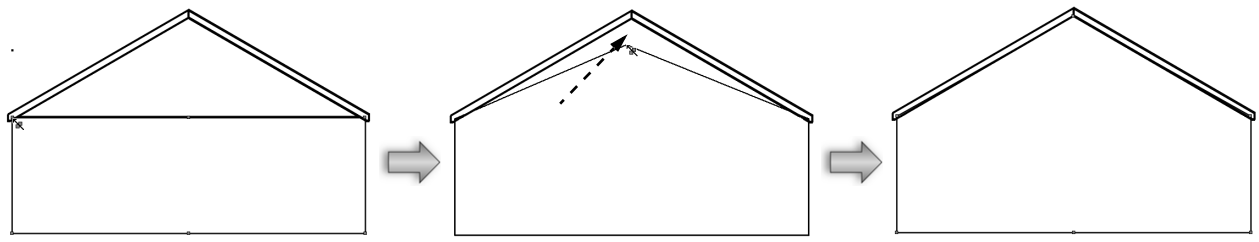
Adding a Vertex to Walls

To add a vertex to a wall:

1. In a 3D view, select the wall that requires a peak (vertex).
2. Click the **Reshape** tool from the Basic palette, and select **Add 3D Wall Peaks** mode.
3. Position the cursor over an end point on one of the corners or an existing vertex, and click.

   When the cursor is over an end point, the standard arrow cursor changes into a single-headed, filled arrow with shaded boxes on either side of the shaft.
4. Move the mouse to add a vertex to the top or bottom of the wall.

   A vertex can be moved to any location along the same wall as long as the location does not pass another existing vertex.
5. Click when the vertex is at the desired location.



With the Add Peak cursor, click-click an end point to add a peak (Front view depicted)

The top of the wall is now flush with the bottom of the roof

To reshape a curved wall to match a planar surface, use the **Subtract Solids** or **Intersect Solids** command with an object that matches the plane of the roof.

Reshaping Walls

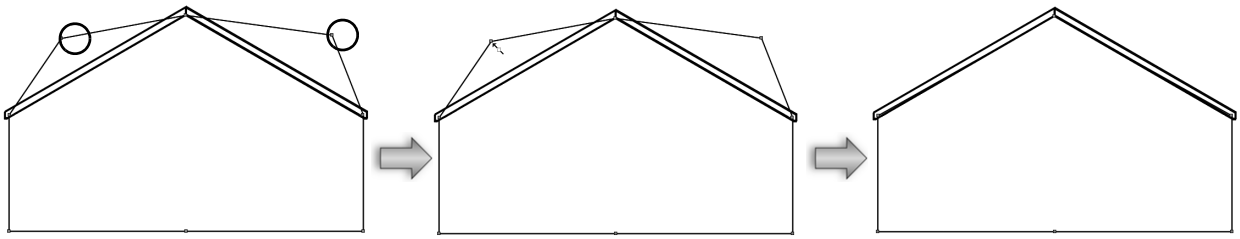
Deleting a Wall Vertex

To delete a control point:

1. In a 3D view, select the wall with the peak (vertex) to be deleted.
2. Click the **Reshape** tool from the Basic palette, and select **Delete 3D Wall Peaks** mode.
3. Position the cursor over the vertex to delete.

   When the cursor is over a vertex, the standard arrow cursor changes into a single-headed, filled arrow with a hollow diamond in the shaft.
4. Click on the vertex.

   The vertex is removed and the wall is reshaped to the remaining vertices.



Wall with two unnecessary vertices (Front view depicted)

With the Delete Peak cursor, click a vertex to delete it

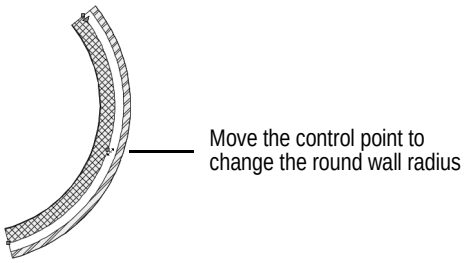
Wall with two vertices removed

Reshaping Walls

Changing Round Wall Radius

To change the radius of a round wall:

1. Select the round wall to edit.
2. Click the **Selection** tool from the Basic palette.
3. Position the cursor over the center control point, and click.
4. Move the mouse to change the radius, and click to set the end point.



Flip the round wall arc by dragging toward, and through, the arc center.

Reshaping Walls

Removing Wall Breaks

The **Remove Wall Breaks** tool cleans up any wall breaks or gaps that were created during editing. For example, when creating a new wall that joins an existing wall, if the new wall is later deleted, a break in the remaining wall displays at the joint. The **Remove Wall Breaks** tool can remove the break and any end caps.

 To remove wall breaks:

1. Click the **Remove Wall Breaks** tool from the appropriate tool set.:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the wall and select **Remove Break** from the context menu.
2. Click and drag to create a marquee box around the wall break or end cap to remove.

   The wall break or end cap is automatically removed.

Reshaping Walls
Joining Walls
Joining Walls with the Fillet Tool

Moving Connected Walls

The **Selection** tool has an additional mode, **Connected Walls**, in the Vectorworks Architect and Landmark products. This mode automatically maintains the connection between the wall being moved and adjoining walls. The involved wall angles remain constrained throughout the move. While dragging a wall, you can enter precise offset distances in the **Offset** field of the Tool bar.

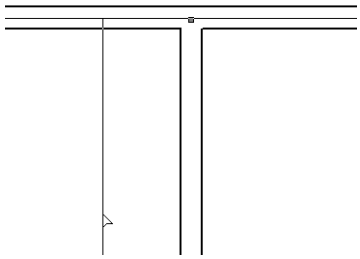
The **Connected Walls** mode does not apply to Y-joined walls or round walls.

Connections between T-joined walls and L-joined walls are also maintained if they are moved using the **Move** or **Nudge** command, or moved and/or duplicated using the **Duplicate Array** command or the **Move by Points** tool, as long as the ends of the moved or duplicated walls overlap with the walls to which they were originally connected.

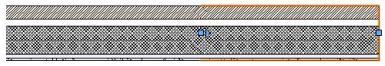
 To move a wall while maintaining its connection to adjacent walls:

1. Click the **Selection** tool from the Basic palette.
2. Select **Connected Walls** from the Tool bar.
3. Click on the desired wall and drag it to a new location; alternatively, change the wall length by editing the **L** value in the Object Info palette (polar coordinates), or by editing a dimension value. The walls stay connected during the move.

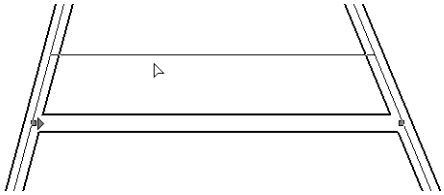
   When you move a wall connected between two other walls, the wall being moved is resized to maintain the connection. A T- or L-joined wall, when moved beyond the end of the connected wall, disconnects the walls.



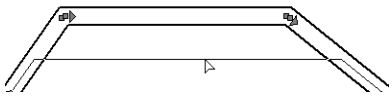
End-to-end walls joined as an L-join move together as a single wall.



For corner joined walls, all involved walls are resized to maintain the connection. A wall cannot be moved in such a way that its length equals zero.



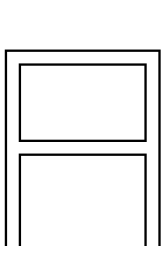
To move a collection of walls without changing the walls relative to each other, deselect the **Connected Walls** mode, and then move the walls.



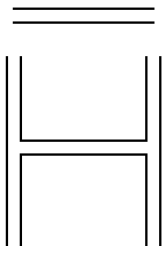
Moving Objects
Duplicate Array
Creating Walls
Creating Wall End Caps
Joining Wall Components
Hiding Wall Components

Automatically Joining Walls

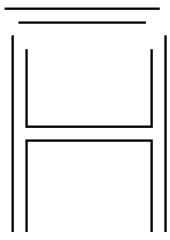
The Vectorworks preference **Auto join walls** automatically joins walls at corners and intersections, and automatically heals the mitered ends of walls when they are separated from one another. For T joins, the break in the side of the wall is healed. Walls can be in either 2D or 3D view.



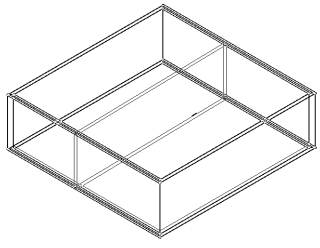
Original walls



Walls separated and ends automatically healed



Walls separated but ends not automatically healed



When drawing walls with auto join on, connected walls are highlighted to show which walls will join

When a core component has been set for walls with components, the components also automatically join. Core components, as well as the other components, join uncapped if they have the same fill, and capped otherwise.

To set the **Auto join walls** preference:

1. Select **Tools > Options > Vectorworks Preferences**.
2. On the Edit tab, select **Auto join walls**.
3. Click **OK**.

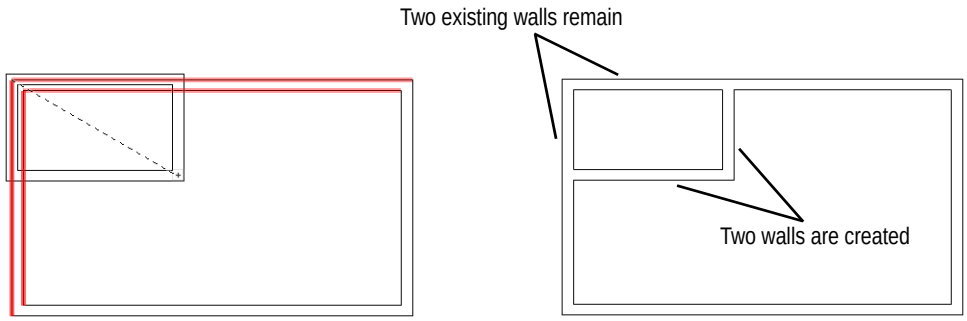
Automatically Joining Walls in Rectangle Mode

When the Vectorworks preference **Auto join walls** is on, walls drawn in **Rectangle** mode that are parallel and overlap or touch each other interact. The drawing operation can be performed using the default Add option, which combines interacting walls, or using the Subtract option, which deletes interacting wall segments. A set of rules determines these interactions.

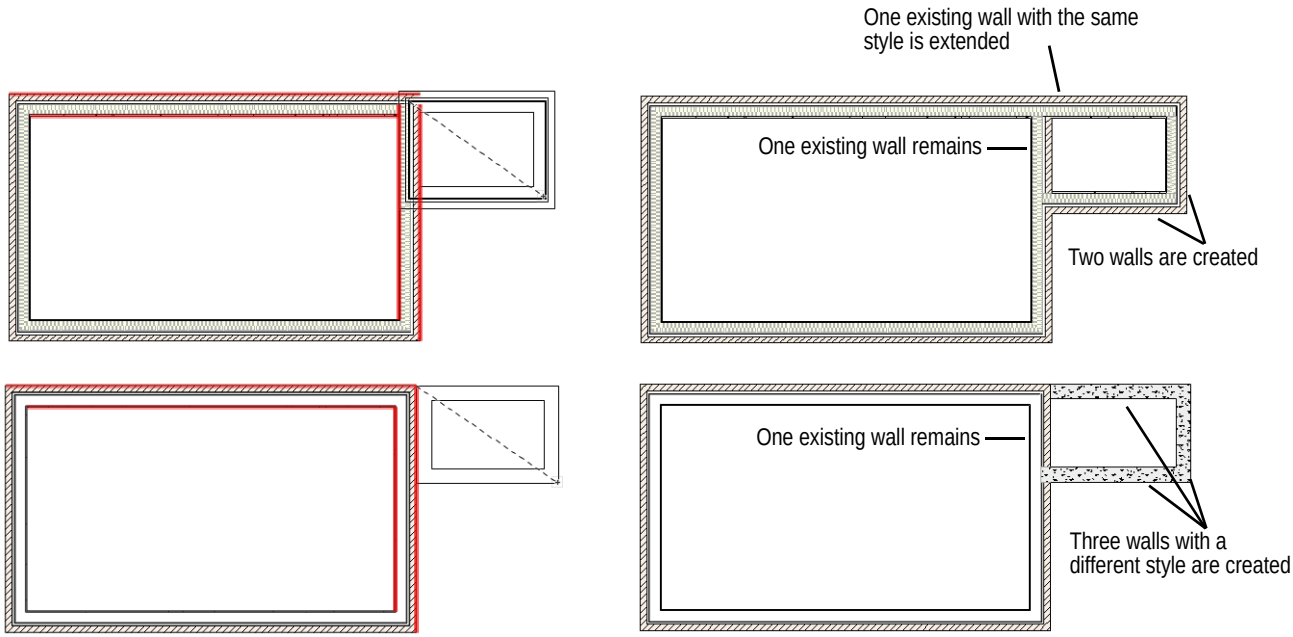
The existing walls that interact with the new walls are highlighted while drawing.

Add option

The existing overlapped wall remains, and the new overlapping wall is not created.

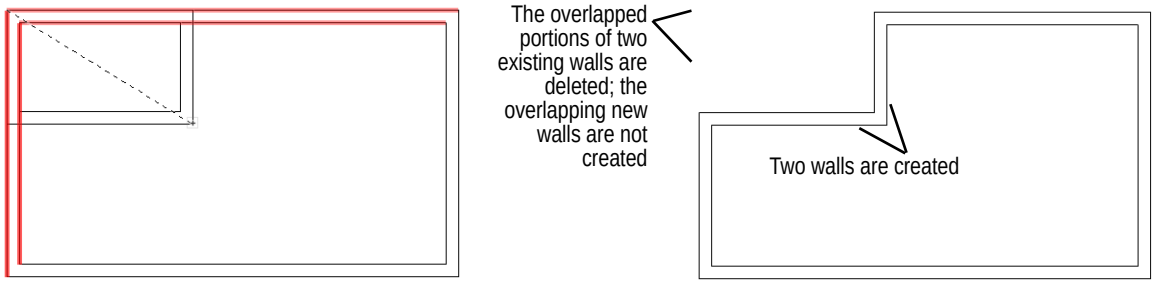


If the new wall extends the existing wall and has the same style, the existing wall is extended and the new wall is not created. If the new wall has a different style, it is created.

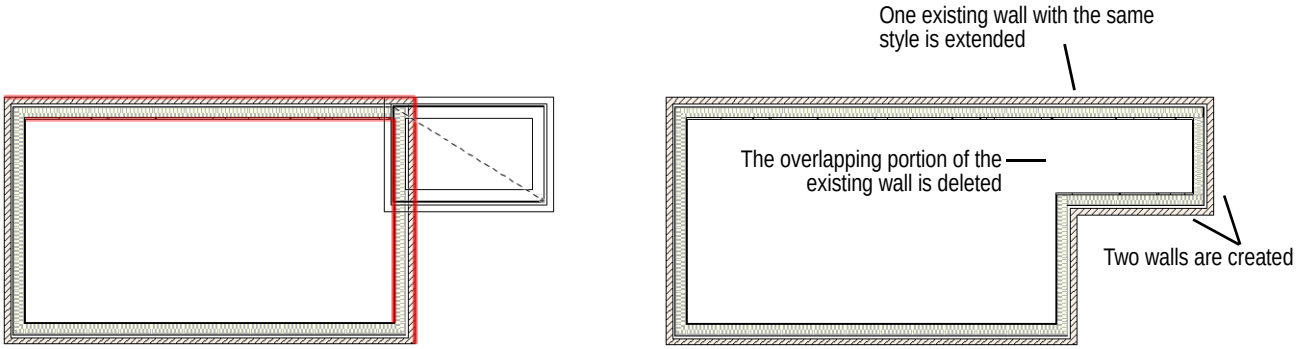


Subtract option–press and hold Alt (Windows) or Option (Mac) while drawing

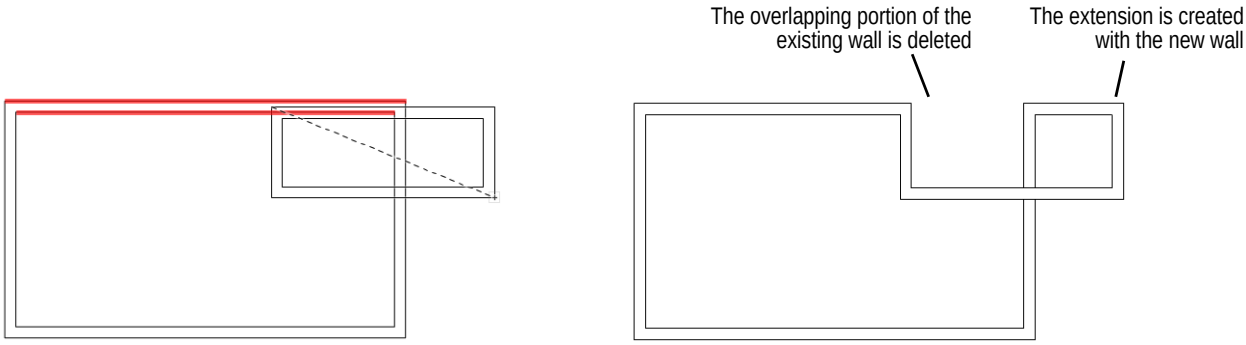
The overlapped portion of the existing wall is deleted, and the new overlapping wall is not created.



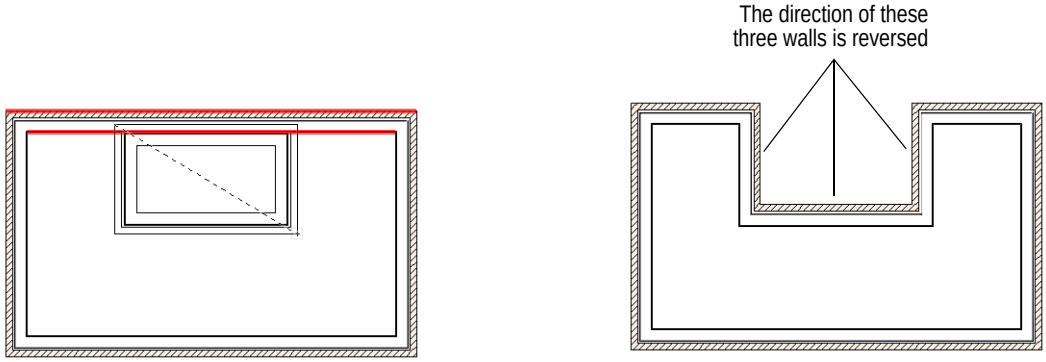
If the new wall extends the existing wall and has the same style, the existing wall is extended and the new wall is not created.



If the new wall partially extends and partially overlaps an existing wall, the overlapping portion is deleted and the extending portion is created with the new wall.



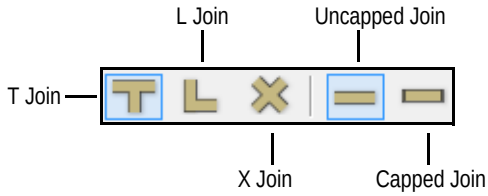
When a wall subtraction requires that the existing walls change direction to maintain the correct orientation, the wall direction changes automatically (see “Wall Direction” on page 506).



Joining Wall Components
Edit Preferences

Joining Walls

The **Wall Join** tool joins straight or curved wall segments not already connected using the **Auto join walls** preference. (See “Automatically Joining Walls” on page 542 for information on **Auto join walls**.) There are three modes for joining walls and two end cap modes.



Mode	Description
T Join	Lengthens or shortens one wall segment until it intersects with the second wall segment selected; creates Y joins by joining the first selected wall to two sections of an existing L join
L Join	Joins the closest ends of two walls to create a corner
X Join	Joins two wall segments at the point where they intersect
Uncapped Join	Applies an uncapped join to wall join operations
Capped Join	Applies a capped join to wall join operations

To temporarily activate this tool, right-click (Windows) or Ctrl-click (Mac) on a wall, and select **Join** from the context menu. The tool defaults to the mode used previously with the tool.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Reshaping Walls
Removing Wall Breaks
T Wall Joins
Y Wall Joins
L Wall Joins
X Wall Joins
Joining Walls with the Fillet Tool
Inserting and Editing Symbols Within a Wall
Automatically Joining Walls

T Wall Joins

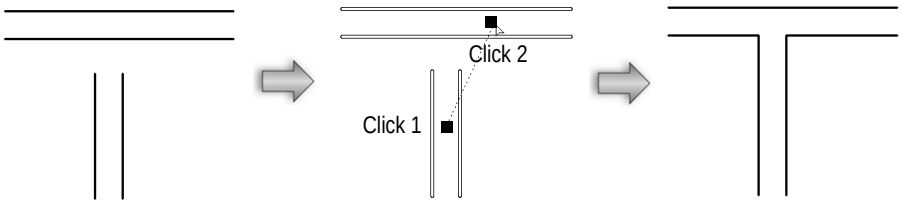
T Join mode lengthens or shortens the first wall segment selected until it intersects with the second wall segment selected. As only the first wall is altered, this mode does not create corner joins.

 To join walls with the T Join mode:

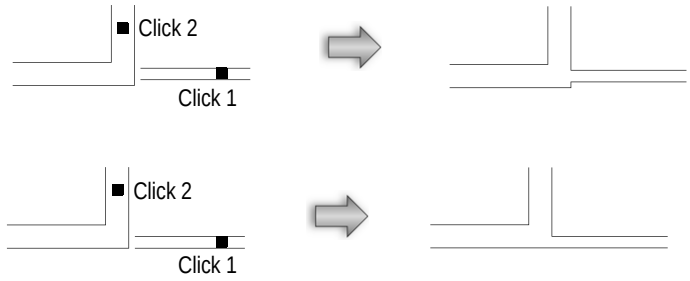
1. Click the **Wall Join** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the first wall to join and select **Join** from the context menu.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Click **T Join** from the Tool bar.
3. Select the wall segment that must change length to intersect with the other wall.
4. Select the second wall segment to join.



To create a T join to an existing corner, join the wall segment to the perpendicular corner segment. This creates a clean join between the walls.



In both cases, two walls are already corner joined; the third wall is joined to the perpendicular segment of the existing corner join.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Joining Walls
Joining Walls with the Fillet Tool

Y Wall Joins

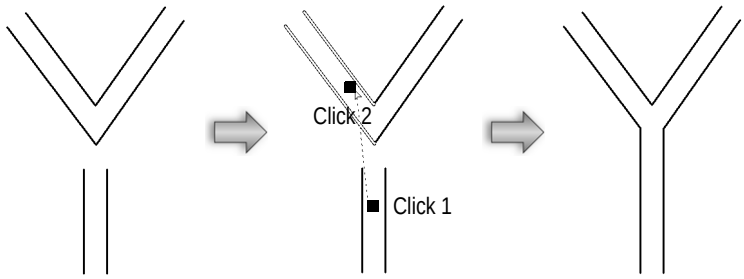
Y wall joins are not automatically created when you use the **Auto join walls** preference. (See “Edit Preferences” on page 49 for information on **Auto join walls**.) Instead, use the T Join mode to create Y wall joins.

To create a Y wall join of three wall segments:

1. Click the **Wall Join** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the first wall to join and select **Join** from the context menu.
2. Click **T Join** from the Tool bar.
3. Select the wall segment that must change length to abut the two previously joined wall segments.
4. Select one of the two wall segments to join.

   The wall segments are joined.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Joining Walls
Joining Walls with the Fillet Tool

L Wall Joins

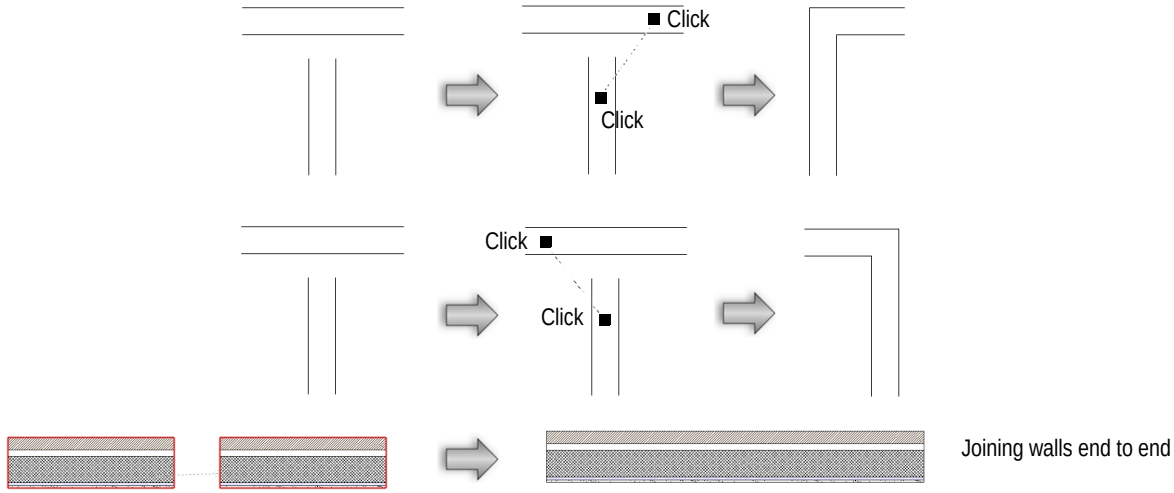
L Join mode joins the closest ends of two walls to create a corner, or joins two walls end to end. Both wall lengths are automatically resized as necessary to meet cleanly.

 To join walls with the L Join mode:

1. Click the **Wall Join** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the wall and select **Join** from the context menu.
2. Click **L Join** from the Tool bar.
3. Select the first wall segment to join.
4. Select the second wall segment to join. For corner joins, the location of the clicks defines the direction of the L.

   The wall lengths are automatically resized and joined.



Joining Walls
Joining Walls with the Fillet Tool

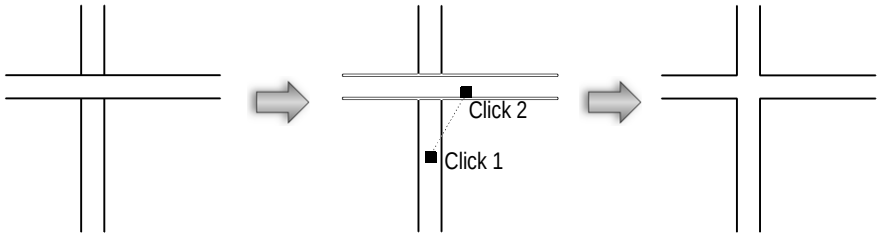
X Wall Joins

X Join mode joins two wall segments at the point where they intersect. Because neither wall segment's length is altered, the two segments must already intersect in order to use this mode.

To join walls with the X Join mode:

1. Click the **Wall Join** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the wall and select **Join** from the context menu.
2. Click **X Join** from the Tool bar.
3. Select the wall segment to become non-load bearing.
4. Select the wall segment to become load bearing.

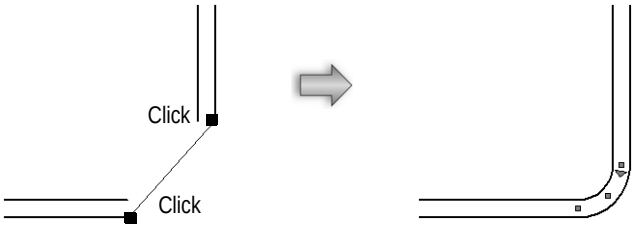


The first wall segment selected is split into two walls segments, which are joined to the load bearing wall.

Joining Walls
Joining Walls with the Fillet Tool

Joining Walls with the Fillet Tool

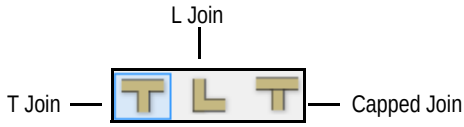
The **Fillet** tool, located on the Basic palette, joins two wall segments by creating a round wall between them. See “Creating Fillets and Chamfers” on page 1077.



Joining Walls

Joining Wall Components

The **Component Join** tool joins the selected components between two straight wall segments; components within pre-existing joined wall segments can also be joined. There are three modes for joining wall components.



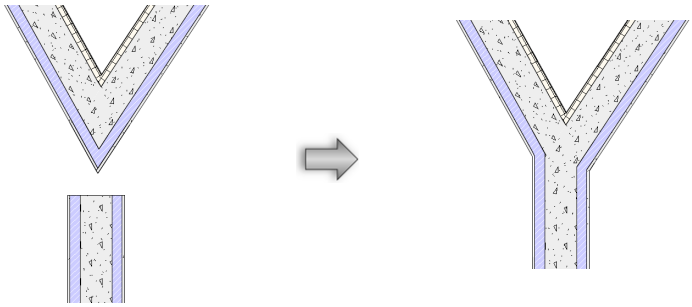
Mode	Description
T Join	Extends or shortens one wall component segment until it intersects with a second wall component segment

Mode	Description
L Join	Joins the closest ends of two wall components to create a corner
Capped Join	Applies a capped join to the component being joined

Components can be joined to the edge of walls instead of specific components in Capped Join mode.

Normally, when you join walls with the **Wall Join** tool, and the Auto join walls preference is enabled, the components automatically join correctly. A core component must be set. (See “Joining Walls” on page 545.)

The **Component Join** tool is useful for adjusting components that did not join as desired when walls were connected, even in complex wall joins.



Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

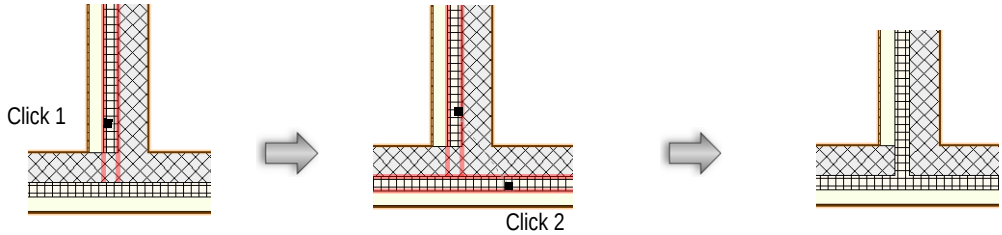
Joining Wall Components
Capped Join Mode
Hiding Wall Components
Joining Walls
Automatically Joining Walls
Creating Walls

T Join Mode

The **T Join** mode extends or shortens one wall component segment until it intersects with a second wall component segment. As only the first component is extended, this mode will not create corner type joins. (For those, use the **L join** mode.)

To join wall components with the **T Join** mode:

1. Click the **Component Join** tool from the Building Shell tool set.
2. Click **T Join** from the Tool bar.
3. Select the component within the wall segment to join. The component is highlighted.
4. Select the second wall segment to join. The component to be joined is highlighted.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

5. Repeat steps 3 and 4 for each component within the wall segment that needs to be joined.

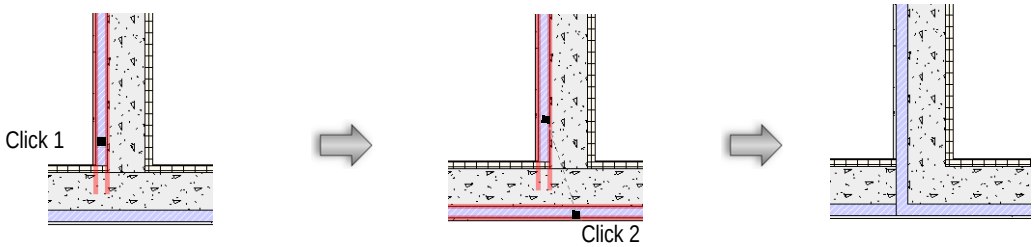
Joining Wall Components
Capped Join Mode
Hiding Wall Components

L Join Mode

The **L Join** mode joins the closest ends of two wall components to create a corner. Both component lengths are extended or shortened, as necessary, until they meet cleanly.

To join wall components with the **L Join** mode:

1. Click the **Component Join** tool from the Building Shell tool set.
2. Click **L Join** from the Tool bar.
3. Select the component within the wall segment to join.
4. Select the second wall segment to join. The component to be joined is highlighted.



5. Repeat steps 3 and 4 for each component within the wall segment that needs to be joined.

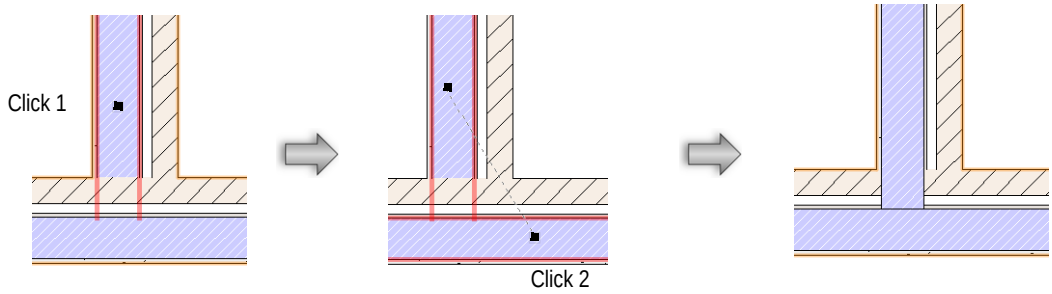
Joining Wall Components
T Join Mode
Capped Join Mode
Hiding Wall Components

Capped Join Mode

Like the **T Join** mode, the **Capped Join** mode extends or shortens one wall component until it intersects with a second component. The component end is capped at the point where it joins the other wall.

To join wall components with the **Capped Join** mode:

1. Click the **Component Join** tool from the Building Shell tool set.
2. Click **Capped Join** from the Tool bar.
3. Select the component within the wall segment to join.
4. Select the second wall segment to join. The component to be joined is highlighted.



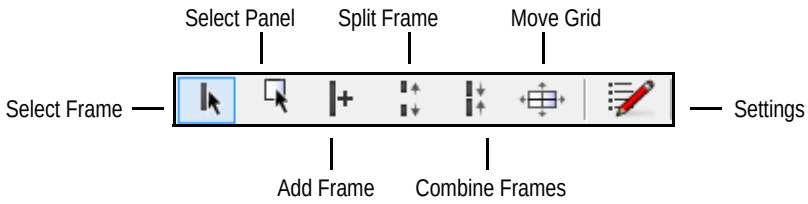
5. Repeat steps 3 and 4 for each component within the wall segment that needs to be joined.

Joining Wall Components
Hiding Wall Components

A L Editing Curtain Walls

The **Edit Curtain Wall** tool manipulates the frames and panels of curtain walls. Once a curtain wall is selected with the tool, the modes allow you to select frames and panels; delete frames and panels; add, split, and combine frames; move frame and panel sets along a wall; and edit frame and panel settings.

When a curtain wall is selected with the **Selection** tool, two context menu commands become available. The **Edit Using Reshape Tool** command activates the **Reshape** tool to reshape the walls as described in “Reshaping Walls” on page 536. The **Edit Using Edit Curtain Wall Tool** command activates the **Edit Curtain Wall** tool.



Mode	Description
Select Frame	Selects one or more frames
Select Panel	Selects one or more panels
Add Frame	Inserts new frames into the curtain wall
Split Frame	Splits a frame into multiple pieces, so that one portion of the frame can be independently moved, edited, or deleted
Combine Frames	Combines two collinear frames into a single frame
Move Grid	Moves an entire set of frames and panels relative to the curtain wall
Settings	Selected frame or panel parameters can be edited

Editing Curtain Wall Frames
Editing Curtain Wall Panels
Adding New Frames to a Curtain Wall
Splitting Curtain Wall Frames
Combining Curtain Wall Frames
Moving a Curtain Wall Grid
Changing Selected Frame or Panel Settings

Creating Curtain Walls

Editing Curtain Wall Frames

To select and manipulate curtain wall frames:

1. Click the **Edit Curtain Wall** tool from the Building Shell tool set, and click **Select Frame** mode from the Tool bar.
2. Select the curtain wall to edit.
3. Select the frame or frames to edit, move, or delete.

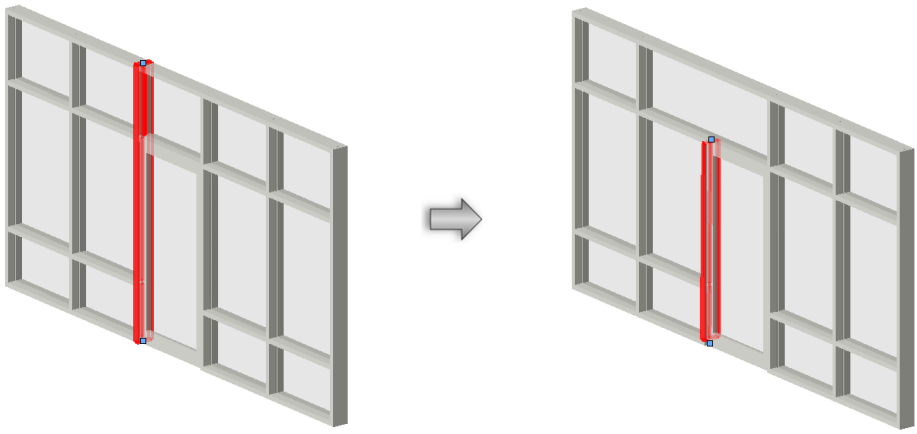
   The **Edit Curtain Wall** tool works similarly to the **Selection** tool. Frames can be selected for editing in a variety of ways.When a single frame is selected, end handles allow the frame to be reshaped.

Method	Selection Action
Click	Standard selection method; selects a single object only
Shift-click	Selects multiple objects as each object is clicked; also can be used to deselect one or more objects without affecting other selected objects
Option-drag (Mac) or Ctrl-drag (Windows)	Places a copy of the object where the mouse button is released
Rectangle marquee	Selects all objects that are completely contained within the marquee
Shift-marquee	Reverses the selection status of objects inside a marquee; if objects inside the marquee are selected, this method deselects those objects
Option-marquee (Mac) or Alt-marquee (Windows)	Selects all objects that the marquee passes through, as well as those contained within the marquee

4. A single, selected frame can be reshaped, moved by dragging, copied, or deleted. The frame must be connected on both ends to another frame. If deleting or moving a frame causes any frames to become disconnected, they are automatically extended until they join a frame.

   Selected frames can be deleted by pressing delete, or alternatively, right-click (Windows) or Ctrl-click (Mac) on the frame and select **Delete Frames** from the context menu.

   Selected frames can be evenly distributed relative to the wall. Frames must be horizontal or vertical, and at least three frames must be selected. Right-click (Windows) or Ctrl-click (Mac) on the frames and select **Distribute Frames** from the context menu. The frames on the ends of the selection do not move.



Panels automatically adjust to conform to the new frame positions.

5. The settings of selected frames can be edited by clicking the **Settings** mode from the Tool bar.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the frame and select **Edit Frames** from the context menu.

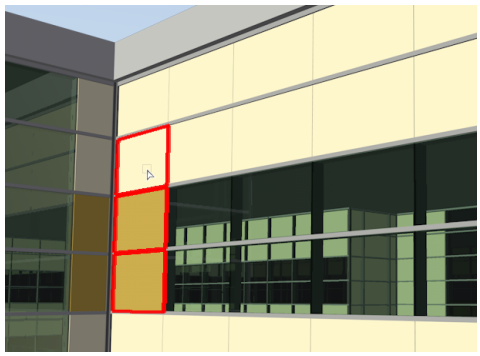
Editing Curtain Walls

Editing Curtain Wall Panels

To select and manipulate curtain wall panels:

1. Click the **Edit Curtain Wall** tool from the Building Shell tool set, and click **Select Panel** mode from the Tool bar.
2. Select the curtain wall to edit.
3. Select the panel or panels to edit. Panels cannot be moved; the position of the panels is determined by the frames. Selecting panels is very similar to selecting frames; see “Editing Curtain Wall Frames” on page 552.
4. The settings of selected panels can be edited by clicking the **Settings** mode from the Tool bar.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the frame and select **Edit Panels** from the context menu.



When one or more curtain wall panels are selected, a door or window can be easily inserted. Right-click (Windows) or Ctrl-click (Mac) on the panel and select **Insert Window** or **Insert Door** from the context menu. The window or door is automatically inserted as a curtain wall window or door, and sized to fit within the selected panel or panels.

Editing Curtain Walls

Adding New Frames to a Curtain Wall

To add new curtain wall frames:

1. Click the **Edit Curtain Wall** tool from the Building Shell tool set, and click **Add Frame** mode from the Tool bar.
2. Select the curtain wall to edit.
3. Click at the start point of the new frame, and then click on the end point of the new frame. The start and end points must be placed on existing frames.

   Press the Shift key to constrain the placement of the second point of the frame horizontally or vertically.
4. The frame is added, acquiring its attributes from the curtain wall grid settings.

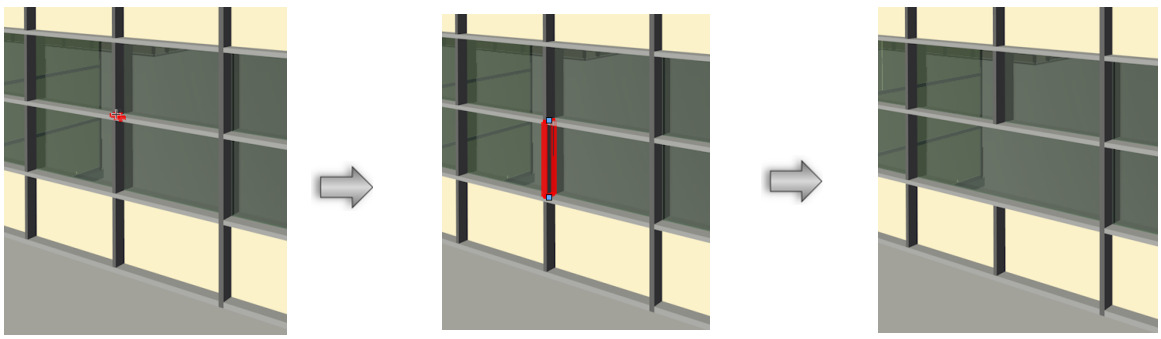
Editing Curtain Walls

Splitting Curtain Wall Frames

To split curtain wall frames:

1. Click the **Edit Curtain Wall** tool from the Building Shell tool set, and click **Split Frame** mode from the Tool bar.
2. Select the curtain wall to edit.
3. Click on a frame to split it into sections.

   A frame can only be split at a frame intersection. A red split cursor displays to indicate locations where the frame can be split.
4. The frame is split, and the resulting sections can be dragged, deleted, or have their settings changed independently.



At an intersection, click on the frame to split | The frame is split into two sections | In this example, one of the split frames was deleted

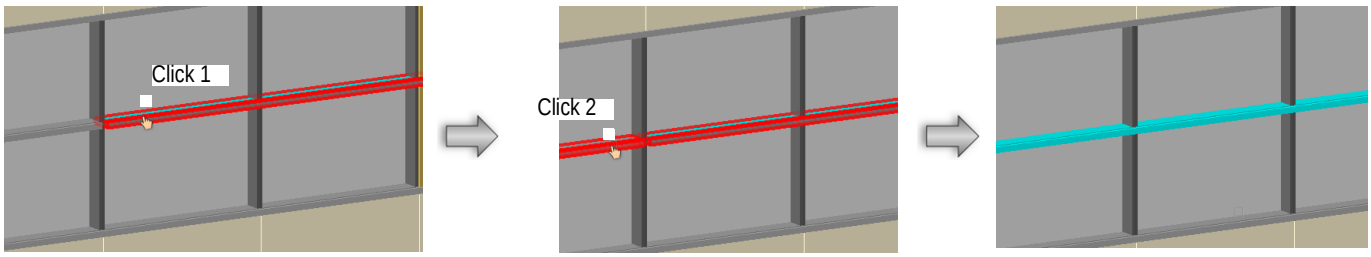
Editing Curtain Walls

Combining Curtain Wall Frames

To combine two collinear curtain wall frames:

1. Click the **Edit Curtain Wall** tool from the Building Shell tool set, and click **Combine Frame** mode from the Tool bar.
2. Select the curtain wall to edit.
3. Click each of the two frames to be combined. Frames must be collinear.

   At a four-way frame intersection, it may be necessary to split a perpendicular frame to be able to join collinear frames.
4. One frame is created. The attributes of the first clicked frame apply to both frames once they are combined.



Click on the first frame | Click on the second frame | The frames are combined

Editing Curtain Walls

Moving a Curtain Wall Grid

The frames and panels of a curtain wall form a grid or pattern relative to the wall. Moving the grid along the wall can be useful to center a grid pattern on a wall.

To move the curtain wall grid relative to the wall:

1. Click the **Edit Curtain Wall** tool from the Building Shell tool set, and click **Move Grid** mode from the Tool bar.
2. Select the curtain wall to edit.
3. Click to set a reference point for the grid.
4. Move the cursor to shift the entire grid of frames and panels, along with any doors and windows, relative to the wall.

   The frames and panels shift to the left or right along the wall. A preview of the grid displays.
5. Click to set the new location of the curtain wall grid.

Click to set a reference point

Move the grid; a preview displays. Click to set the new grid position.

The grid is shifted relative to the wall

Editing Curtain Walls

Changing Selected Frame or Panel Settings

The **Settings** mode changes the settings of either frames or panels that have been selected with the **Select Frame** or **Select Panel** mode of the **Edit Curtain Wall** tool.

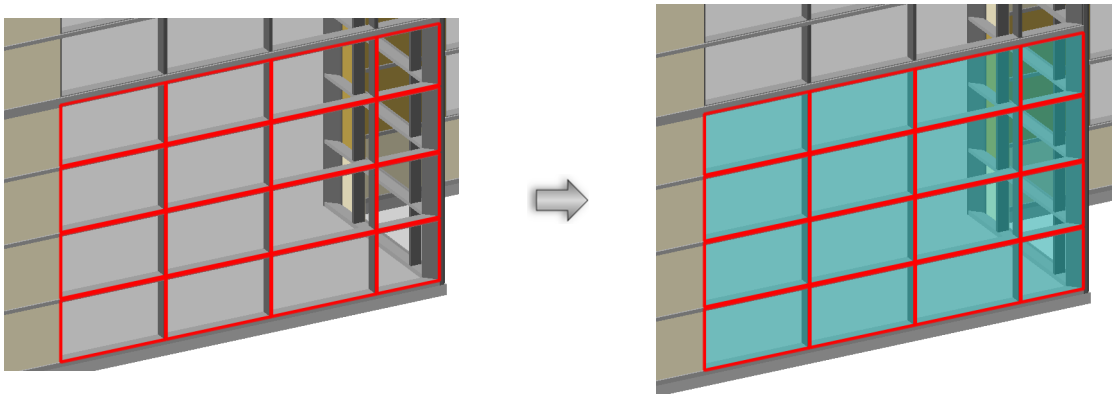
To change curtain wall frame or panel settings:

1. Click the **Edit Curtain Wall** tool from the Building Shell tool set.
2. Select the curtain wall to edit.
3. Using either **Select Frame** mode or **Select Panel** mode of the **Edit Curtain Wall** tool, select the frames or panels to edit.
4. Click **Settings** mode from the Tool bar.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the frame or panel and select **Edit Frames** or **Edit Panels** from the context menu.
5. The Frame Settings or Panel Settings dialog box opens. Change the frame or panel settings as described in “Setting Curtain Wall Frame Parameters” on page 526 or “Setting Curtain Wall Panel Parameters” on page 528.

   The Frame Settings dialog box, when opened in this way, does not include the frame category selection on the left, since the frames to edit have already been selected.
6. Click **OK**.

   The frame or panel settings are changed.



Editing Curtain Walls
Editing Curtain Wall Frames
Editing Curtain Wall Panels

Creating Columns and Pilasters

The Vectorworks Fundamentals product and all Vectorworks Design Series products include a basic architectural column object (for aesthetic and space-defining purposes). In the Architect product, the column object offers additional structural capabilities, and a pilaster object is provided.

In the Architect product, columns and pilasters can be architectural in nature, structural in nature, or both. Use the **Column** tool or **Pilaster** tool to draw an object that not only reflects the architectural appearance of a column/pilaster, but also defines it in structural terms that can be used in an engineering analysis of the building. Structural and architectural elements can be displayed or hidden, as desired. The column ID can also, optionally, be placed on the drawing. Additionally, in the Architect product, the column/pilaster can be exported to IFC format for exchange with structural analysis programs.

A pilaster can be placed in a wall (however, a column cannot be placed in a wall). A pilaster inserted in a wall can set its architectural height relative to the wall.

Tool access varies depending on the Vectorworks product installed:

Vectorworks Product	Tool Set / Tool
Fundamentals	Walls > Column (basic column)
Architect and Designer	Building Shell > Column (architectural column with additional capabilities)
Architect and Designer	Building Shell > Pilaster
Landmark and Spotlight	Building Shell > Column (basic column)

 To create a column:

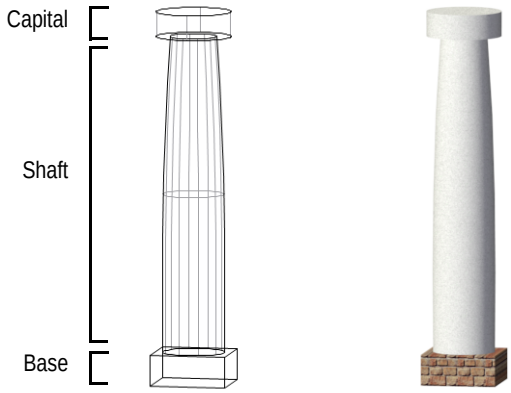
 To create a pilaster:

1. Click the **Column** tool or the **Pilaster** tool from the appropriate tool set:
   - Fundamentals workspace: Walls tool set
   - Design Series workspaces: Building Shell tool set
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time a column/pilaster is placed in the drawing, the object properties dialog box opens. These parameters apply to subsequently created columns/pilasters; they can be changed later by accessing them from the Object Info palette.
3. Specify the column/pilaster properties and click **OK**.

   To create a circular or square shaft, capital, or base, enter the same value for the width and depth.

Basic Column Parameters

The basic column object is available with the Vectorworks Fundamentals, Landmark, and Spotlight products.



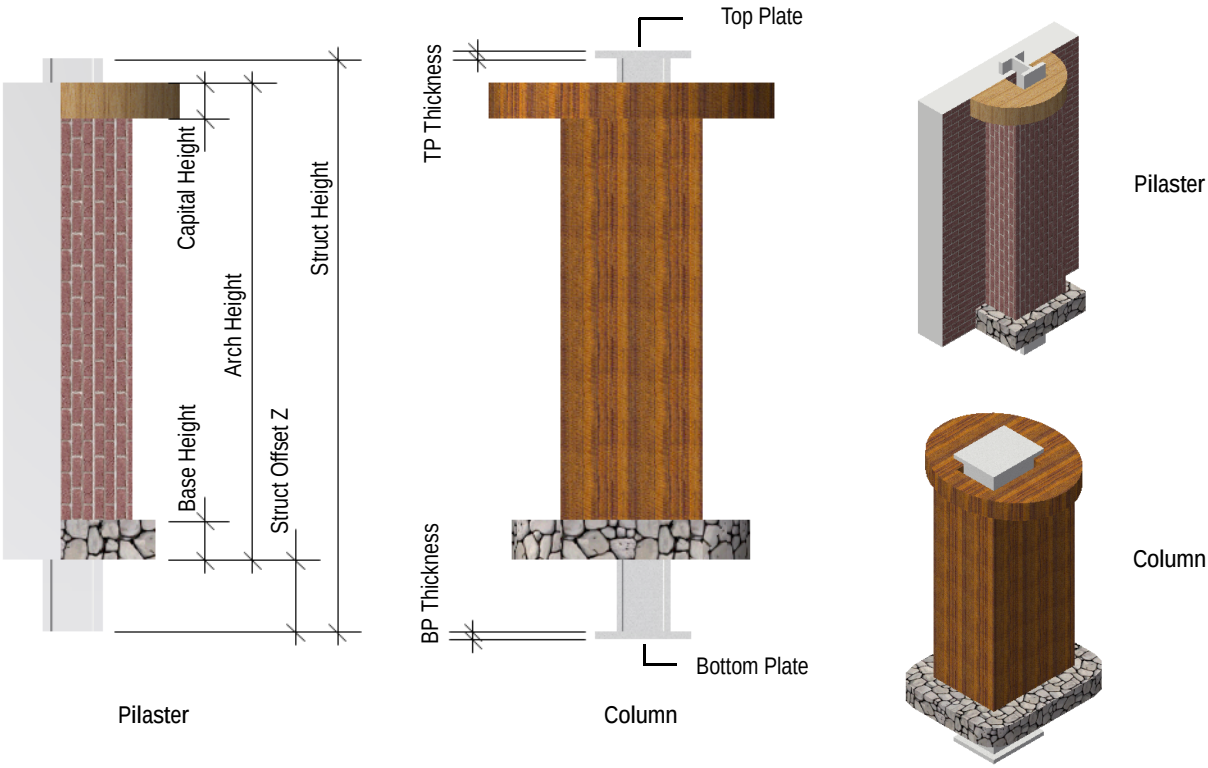
Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Use Center Marks	Select this option to size the cross at the column insertion point and enter the **Center Mark Size**
Height	Directly sets the desired height of the column, including base and capital (if any). When the column height is determined manually by this method, the **Top Bound** property of the column is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the column is bound by the layer wall height value, the wall height updates automatically.
Top Bound	Sets the vertical reference that determines the top of the column. The **Layer Wall Height** value is set by the design layer (see "Setting Design Layer Properties" on page 165) Additional options are available for the Vectorworks Architect product.
Top Offset	Sets the offset of the top of the column from its specified top bound height. If setting the boundary of the top of the column to the layer elevation, enter the column's height for the **Top Offset**; if setting the height of the top of the column to the default wall height, specify any offset above or below the layer's default wall height.
Bot Bound	Sets the vertical reference that determines the bottom of the column; **Layer Elevation** is the only option available unless the Vectorworks Architect product is installed
Bot Offset	For the bottom of the column, sets the offset from the layer elevation
Shaft Type	Select to draw the column with an oval or rectangular shaft type
Shaft Width	Enter the shaft width
Shaft Depth	Enter the shaft depth
Shaft Corner Radius	For rectangular shaft type, enter the corner radius of the shaft

Parameter	Description
Shaft Taper	Select whether to draw the column with a classic or straight taper, or no taper
Taper Width	For a straight or classic taper, enter the taper width
Taper Depth	For a straight or classic taper, enter the taper depth
Use Capital	Select whether to add a capital to the column
Capital Type	Select to draw the column with an oval, oval mushroom, rectangular, or rectangular mushroom capital
Capital Width	Enter the capital width
Capital Depth	Enter the capital depth
Capital Height	Enter the capital height
Capital Corner Radius	For rectangular capital type, enter the corner radius of the capital
Use Base	Select whether to add a base to the column
Base Type	Select an oval or rectangular base type
Base Width	Enter the base width
Base Depth	Enter the base depth
Base Height	Enter the base height
Base Corner Radius	For rectangular base type, enter the base corner radius
Base Divisions	Enter the number of divisions for the base
Divider Depth/Height	For more than one base division, enter the depth and height for each divider
Get Finishes/Classes	Click to open the **Get Finishes/Classes** dialog box to select the class to use for rendering the 3D shaft, capital, and base of the column
Shaft Finish	Displays the class to use for rendering the 3D shaft (selected in the Get Finishes/Classes dialog box)
Capital Finish	If **Use Capital** is selected, displays the class to use for rendering the 3D capital (selected in the Get Finishes/Classes dialog box)
Base Finish	If **Use Base** is selected, displays the class to use for rendering the 3D base (selected in the Get Finishes/Classes dialog box)

A Architect Column and Pilaster Parameters

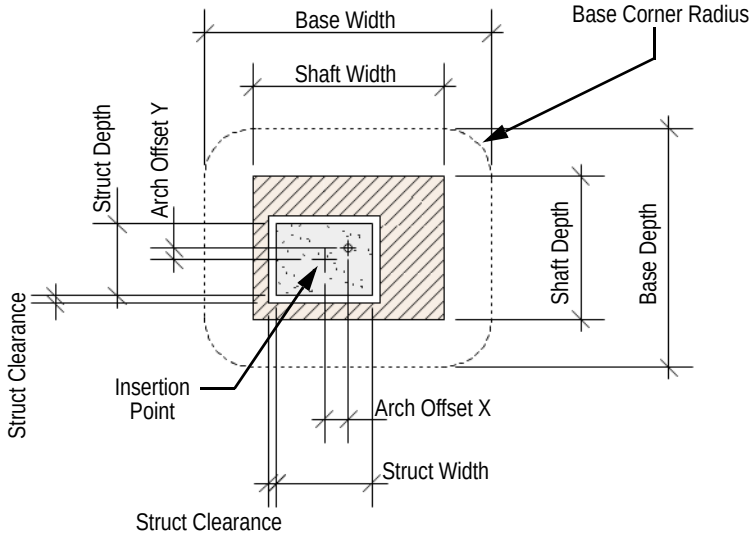
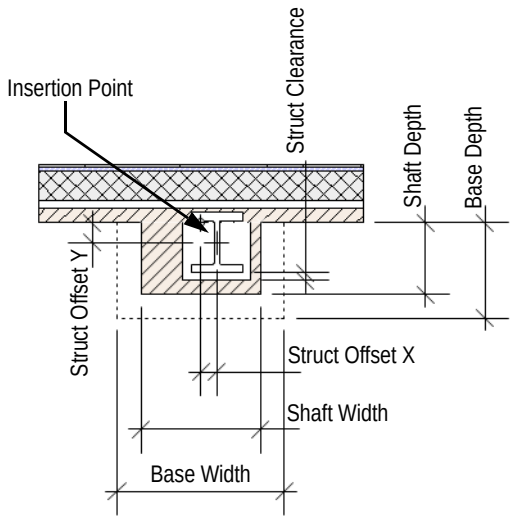
The architect column and pilaster objects are available with the Vectorworks Architect and Designer products.



Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Use Center Marks	Select this option to size the cross at the column/pilaster insertion point and enter the **Center Mark Size**
Use Component	Select whether to use only architectural components, only structural components, or a combination of architectural and structural components for the column/pilaster. Depending on the selection, different options are available. When both Architectural and Structural components are selected, the height of the architectural and structural components can be set individually.
Use Wall Height (Pilaster only, inserted in wall)	For architectural only or architectural and structural components, select to draw the architectural pilaster height equal to the wall height. This option is only available when a pilaster has been inserted into a wall.
Height (Architectural Only or Structural Only)	Directly sets the desired height of the column/pilaster's architectural or structural component, including base and capital (if any). When the height is determined manually by this method, the **Top Bound** property of the column/pilaster is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the column/pilaster is bound by the layer wall height value or by a story level, the column/pilaster height updates automatically.

Parameter	Description
Arch Height (Architectural and Structural)	When architectural components are included with structural, directly sets the desired height of the column/pilaster's architectural component, including base and capital (if any). When the height is determined manually by this method, the **Top Bound** property of the column/pilaster is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the column/pilaster is bound by the layer wall height value or by a story level, the column/pilaster height updates automatically.
Struct Height (Architectural and Structural)	When structural components are included with architectural, directly sets the desired height of the column/pilaster's structural component, including top plate and base plate (if any). When the height is determined manually by this method, the **Top Bound** property of the column/pilaster is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the column/pilaster is bound by the layer wall height value or by a story level, the column/pilaster height updates automatically.
Top Bound	Sets the vertical reference that determines the top of the object. The **Layer Wall Height** value is set by the design layer (see "Setting Design Layer Properties" on page 165) Additional options are available for the Vectorworks Architect product. The top of the object can be bound by one of the story levels defined for the story or the story above it. By setting the top of the object to a level type, if the elevation of the associated story changes, the height of the object changes automatically to match.
Top Offset	Sets the offset of the top of the object from its specified top bound height
Bot Bound	Sets the vertical reference that determines the bottom of the object; **Layer Elevation** is the only option available unless the Vectorworks Architect product is installed. Additional options are available for the Vectorworks Architect product. The bottom of the object can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the object to a level type, if the elevation of the associated story changes, the height of the object changes automatically to match.
Bot Offset	For the bottom of the object, sets the offset from its specified bottom bound height
Arch. Comp. Class	For Architectural Only or Architectural and Structural components, displays the class for the architectural components of the column/pilaster (selected in the Get Finishes/ Classes dialog box)

Parameter	Description
Arch Offset X / Arch Offset Y (Column only)	For Architectural and Structural components, specify the distance to offset the architectural component of the column from the insertion point, in the X and Y directions The 2D column's insertion point always coincides with the center marks (if any) and the center of the structural component (if any), and the architectural component is offset from the insertion point as specified by these properties 
Struct Offset X / Struct Offset Y (Pilaster only)	For Architectural and Structural components, specify the distance to offset the structural component of the pilaster from the insertion point, in the X and Y directions The 2D pilaster's insertion point always coincides with the face of the wall, centered on the architectural component; the structural component and center marks (if any) are offset from the insertion point, as specified by these properties 
Struct Offset Z	For Architectural and Structural components, specify the distance to offset the structural component of the column/pilaster from the architectural component, in the Z direction

Parameter	Description
Shaft Type	For Architectural Only or Architectural and Structural components, select to draw the column/pilaster with an oval or rectangular shaft type
Shaft Width	For Architectural Only or Architectural and Structural components, enter the shaft width
Shaft Depth	For Architectural Only or Architectural and Structural components, enter the shaft depth
Shaft Corner Radius	For Architectural Only or Architectural and Structural components, enter the corner radius for a rectangular shaft type
Shaft Taper	For Architectural Only or Architectural and Structural components, select whether to draw the column/pilaster with a classic or straight taper, or no taper
Taper Width	For a straight or classic taper, enter the taper width
Taper Depth	For a straight or classic taper, enter the taper depth
Shaft Finish	For Architectural Only or Architectural and Structural components, displays the class to use for rendering the 3D shaft (selected in the Get Finishes/Classes dialog box)
Use Capital	For Architectural Only or Architectural and Structural components, select whether to add a capital to the architectural component
Capital Type	Select to draw the column/pilaster with an oval, oval mushroom, rectangular, or rectangular mushroom capital
Capital Width	Enter the capital width
Capital Depth	Enter the capital depth
Capital Height	Enter the capital height
Capital Corner Radius	For rectangular capital type, enter the corner radius of the capital
Capital Finish	For Architectural Only or Architectural and Structural components, displays the class to use for rendering the 3D capital (selected in the Get Finishes/Classes dialog box)
Use Base	For Architectural Only or Architectural and Structural components, select whether to add a base to the architectural component
Base Type	Select an oval or rectangular base type
Base Width	Enter the base width
Base Depth	Enter the base depth
Base Height	Enter the base height
Base Corner Radius	For rectangular base type, enter the base corner radius
Base Divisions	Enter the number of divisions for the base
Divider Depth / Height	For more than one base division, enter the depth and height for each divider
Base Finish	For Architectural Only or Architectural and Structural components, displays the class to use for rendering the base (selected in the Get Finishes/Classes dialog box)
Column ID	For Structural Only or Architectural and Structural components, enter the **Column ID** (for reporting purposes, and optionally to be displayed in the drawing)

Parameter	Description
Display Column ID	Select this option to display the column ID on the drawing; the ID text is placed at the insertion point of the column/pilaster, at the center of a control point. Click and drag the control point to move the ID to the desired location on the drawing. The ID text attributes can be modified using the **Text** menu commands.
Column ID Class	Displays the class to use for the column ID text (selected in the Get Finishes/Classes dialog box)
Get Finishes/Classes	Click to open the **Get Finishes/Classes** dialog box to select the class to use for attributes of the 3D shaft, capital, and base of the column and to select the class for the column ID text (if applicable) and the architectural and/or structural components of the column/pilaster (as applicable, based on the component type specified); in addition to classes present in the drawing, architectural component classes include the Wall-Interior and Structural-Columns class standards and structural component classes include the Structural-Columns class standard. Alternatively, create a new class by selecting New, or select the class named <Object Class> which places the item in the same class as the column or pilaster object.
Struct Type	For Structural Only or Architectural and Structural components, select whether the structural component is made of concrete, wood, or steel
Struct Shape	For Structural Only or Architectural and Structural components, specify whether the concrete or wood structural shape is oval or rectangular
Struct Width	For Structural Only or Architectural and Structural components, enter the width of the concrete or wood structural component
Struct Depth	For Structural Only or Architectural and Structural components, enter the depth of the concrete or wood structural component
Struct Corner Radius	For Structural Only or Architectural and Structural components, enter the structural corner radius for a rectangular structural shape
Select Structural Shape	For Structural Only or Architectural and Structural components, click to open the Select Structural Shape dialog box to select the structural shape to use for steel structural types. Select the structural shape from the default content or the current file's content (see "Resource Libraries" on page 219). Select the shape's series and size, and click **OK**.
Struct Shape / Struct Series / Struct Size	For Structural Only or Architectural and Structural components, displays the selected steel structural shape symbol name, series, and size
Struct Height	For Structural Only or Architectural and Structural components, specify the overall height of the structural column/pilaster, including base plate and top plate (if any)
Struct Rotation	For Structural Only or Architectural and Structural components, enter the number of degrees to rotate the structural component
Struct Clearance	For Architectural and Structural components, specifies the clearance around the structural component that will be subtracted from the architectural component
Struct. Comp. Class	For Structural Only or Architectural and Structural components, displays the class for the structural component of the column/pilaster (selected in the Get Finishes/Classes dialog box)
Use Top Plate	For Structural Only or Architectural and Structural components, select whether to add a plate to the top of the structural component

Parameter	Description
TP Shape	Select to draw either an oval or rectangular top plate
TP Width	Enter the width of the top plate
TP Depth	Enter the depth of the top plate
TP Thickness	Enter the thickness of the top plate
TP Corner Radius	For a rectangular top plate, enter the corner radius
Use Base Plate	For Structural Only or Architectural and Structural components, select whether to add a plate to the bottom of the structural component
BP Shape	Select to draw either an oval or rectangular base plate
BP Width	Enter the width of the base plate
BP Depth	Enter the depth of the base plate
BP Thickness	Enter the thickness of the base plate
BP Corner Radius	For a rectangular base plate, enter the corner radius

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Exporting Columns and Pilasters to IFC Format

Exporting Columns and Pilasters to IFC Format

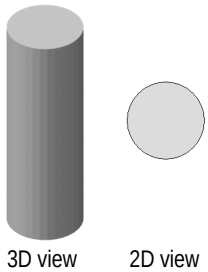
In the Vectorworks Architect product, columns and pilasters can be exported to IFC format for exchange with structural analysis programs. If the column/pilaster has both architectural and structural components enabled, the object is exported as an IfcColumn and the architectural component is exported as an IfcCovering with an IfcCoveringType of Cladding. If the column/pilaster has only an architectural or structural component enabled, the object is exported as an IfcColumn.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Columns and Pilasters
IFC Format Interoperability
GSA Spatial Program BIM Requirements

Creating Pillars

The **Pillar** command converts any closed 2D shape—rectangle, circle, oval, or polygon—into a pillar. In addition, use it on open 2D shapes, such as lines and polylines, to create a flat, screen-like object. These objects include such things as movie screens, room dividers, and moving walls. Once created, a pillar can be joined to a wall.



To create a pillar:

1. Click on the object to convert.
2. Select the **Pillar** command from the appropriate menu:

- Fundamentals workspace: **Model > AEC > Pillar**
- Architect workspace: **AEC** > **Pillar**
- Landmark workspace: **Landmark > Architectural > Pillar**
- Spotlight workspace: **Spotlight > Architectural > Pillar**

The Pillar Preferences dialog box opens.

3. Enter a pillar height.
4. Click **OK**.

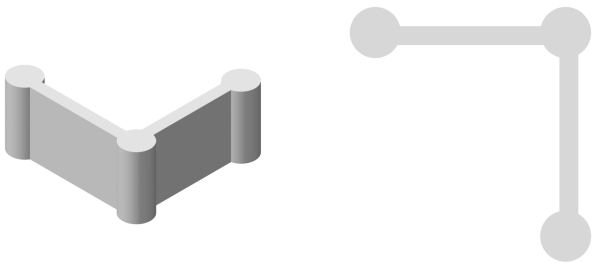
   After creation, the pillar can be edited by selecting **Modify > Edit Pillar**. See “Object Editing Mode” on page 1004.

Joining Pillars and Walls

Joining Pillars and Walls

Walls and pillars can be joined together. Any number of walls can connect to the pillar as long as space exists on the pillar.

The pillar needs to be joined to the end of a wall; it cannot be added to an existing wall intersection such as the corner of L-joined or T-joined walls.



3D view 2D view

To join a pillar to a wall:

1. Click on the pillar.
2. Draw the walls.

   If the Auto-join feature is enabled, then the walls automatically connect to the pillar.

   If the Auto-join feature is disabled, then click the **Wall Join** tool from the Walls tool set or the Building Shell tool set and join the walls to the pillar.

Creating Pillars
Automatically Joining Walls

Roofs

There are two ways to create a roof for a structure:

- The **Roof Face** command converts any closed 2D object into a roof face object.
- The **Create Roof** command creates a roof object by selecting the object(s) that the roof will be based upon (including walls, polygons and polylines).

Use the **Roof Face** command to create hybrid (2D/3D) roof structures from any closed 2D object. This is an easy way to create a slab or flat roof. You can also use this command to create a uniquely shaped roof, such as a gable roof over a round structure. Roof faces created with this command can be connected to other roof faces; roof faces can contain elements like gables and skylights.

The **Create Roof** command creates a roof object with multiple faces based on specified parameters. A great variety of roof types can be created by this method, and they can include roof elements such as gables and skylights. Once the walls of a structure are complete, select the walls that the roof will be based upon, and specify the roof parameters. Alternatively, select a polygon or polyline as the basis for the roof.

The roof can be created in the same design layer as the wall structure, or in a different layer. If the walls and roof are on different layers, create a viewport to view the walls and roof together.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Roof Faces

To create a roof face:

1. Draw the 2D object that is the basis for the roof. Any enclosed 2D object can be used, such as enclosed arcs, polylines, rectangles, circles, and ovals.
2. With the object selected, select the **Roof Face** command from the appropriate menu:
   - Fundamentals workspace: **Model > AEC > Roof Face**
   - Architect workspace: **AEC** > **Roof Face**
   - Landmark workspace: **Landmark > Architectural > Roof Face**
   - Spotlight workspace: **Spotlight > Architectural > Roof Face**

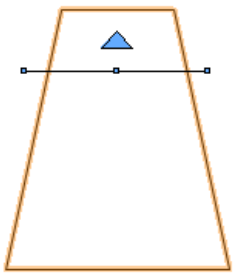
   The Create Roof Face dialog box opens. Specify the roof slope creation method, edge and hole miter options, and roof parameters.

Click to show/hide the parameters.

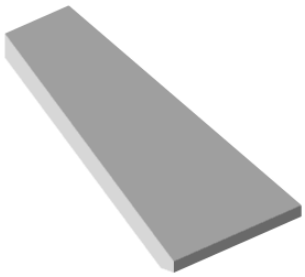
Parameter	Description
Roof Slope	Indicates the roof slope creation method and criteria
Angle	Creates a roof slope based on an angle; specify the **Angle**
Rise over Run	Creates a roof slope based on rise over run values; specify the **Rise** and **Run**. The rise is the distance along the Y axis the roof rises above the Z height, and run is the distance along the X axis for the roof to reach that height.
2nd Click Height	Creates a roof slope based on a mouse click position (this option only available in Top/Plan view). Enter the **Height** for the second mouse click.
Edge Miter	Indicates the miter type for the roof edge
Vertical	Creates the roof edge perpendicular to the active layer plane
Horizontal	Creates the roof edge parallel to the active layer plane
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Double	Creates the roof edge with a horizontal and vertical miter; specify the **Horizontal** and **Vertical** lengths
Square	Creates the lower edge of the roof face perpendicular to the roof surface, regardless of the roof angle
Hole Miter	Specifies miter options for a roof cutout
Vertical	Cutout edges are perpendicular to the active layer plane
Splayed	The bottom edge of a cutout is perpendicular to the active layer plane, and the top edge of a cutout of is parallel to the active layer plane
Square	Cutout edges are perpendicular to the roof surface
Axis Z	Sets the roof height at the axis
Angle	For angled roof slopes, indicates the roof slope angle
Rise/Run	For rise over run roof slopes, specifies the rise and run values
Height	Indicates the height of the second mouse click when the roof slope is specified with that option
Thickness	Specifies the roof thickness
Vertical/Horizontal	For double miters, specifies the horizontal and vertical miter lengths

3. Click **OK**.
4. Click-drag the mouse across the 2D object to draw a line that defines the roof axis; click again to complete the line. An arrow displays on one side of the line, indicating which side of the object will be the high side of the roof.
5. Point the mouse to the side of the roof that will be highest, and then click again. The roof axis line displays selection handles, and the arrow pointing to the high side of the roof turns blue, indicating that the roof face is complete.



2D view



3D view

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing and Reshaping Roof Faces
Connecting Roof Faces
Creating Cutouts in a Roof Face
Adding Roof Elements to Roof Objects and Roof Faces
Roofs
Applying a Texture to an Object

Editing and Reshaping Roof Faces

Once a roof face has been created, there are several ways to edit it:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Edit the roof face parameters (such as the angle or eave type) in the Object Info palette.
- Change the roof's basic 2D object shape using the **Edit Roof** command. (Be careful not to use the **Ungroup** command, as this permanently changes the roof face back into the original 2D object.) Use the **Exit Roof** command to complete the change.
- Move the roof face axis using the **Selection** tool in Top/Plan view.
- Change the roof face angle using the **Reshape** tool (best in Front or Back view).

  The roof angle must be between 0° and 85° to be changed by the **Reshape** tool.
- Add gable windows or skylights as described in "Adding Roof Elements to Roof Objects and Roof Faces" on page 581.

The roof face properties can be edited from the Object Info palette. The roof face parameters are described in "Creating Roof Faces" on page 567. Only the parameters which are different are described here.

Click to show/hide the parameters.

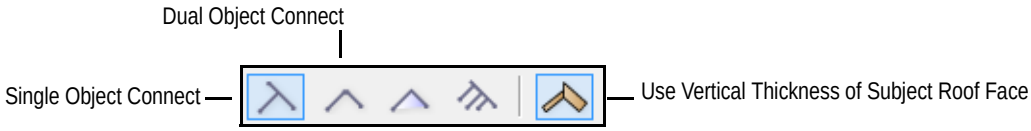
Parameter	Description
Dormer Settings	When the roof face includes one or more dormers, their parameters can be set. Parameter settings affect all dormers in the roof face.
Wall Thickness	Sets the thickness of the walls in the dormer
Roof Thickness	Sets the thickness of the dormer roof

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing a Group
Reshaping Objects
Applying a Texture to an Object

Connecting Roof Faces

Use either the Single Object Connect or the Dual Object Connect mode of the **Connect/Combine** tool to trim roof faces that intersect or to extend one roof face to another. If the roof faces have different thicknesses, there is an option to adjust the thicknesses to match automatically.

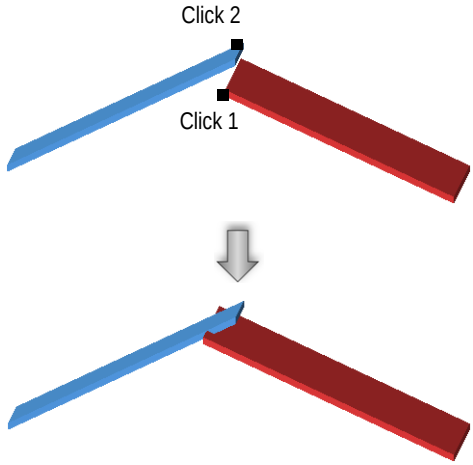


To connect two roof faces:

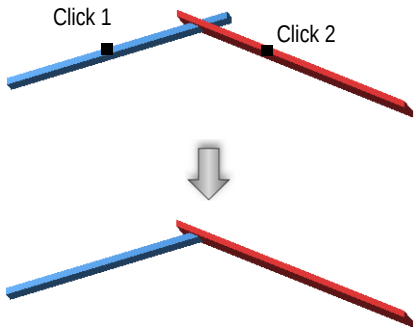
1. Click the **Connect/Combine** tool from the Basic palette, and select either **Single Object Connect** or **Dual Object Connect** from the Tool bar.
2. If the roof faces have different thicknesses, set the **Use Vertical Thickness of Subject Roof Face** option on the Tool bar as appropriate.
   - To automatically adjust the roof face thickness of the second clicked roof face to match the vertical thickness of the first clicked roof face, select **Use Vertical Thickness of Subject Roof Face**.
   - To connect the roof faces without adjusting the thickness, de-select the option.
3. Click the first, and then the second, roof face to connect.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The result of the operation depends on the mode selected, the positions of the roof faces, and the portion of the roof faces clicked upon.

Single Object Connect mode

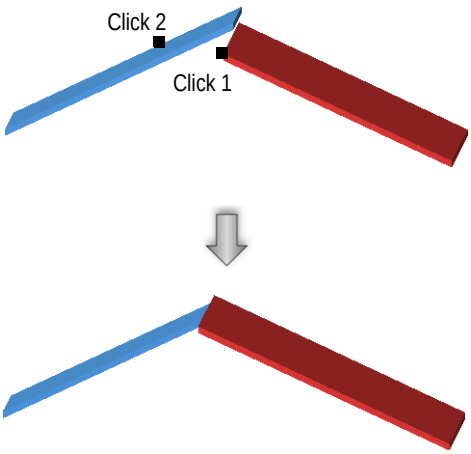


For non-intersecting faces, the first clicked face is extended to meet the second clicked face

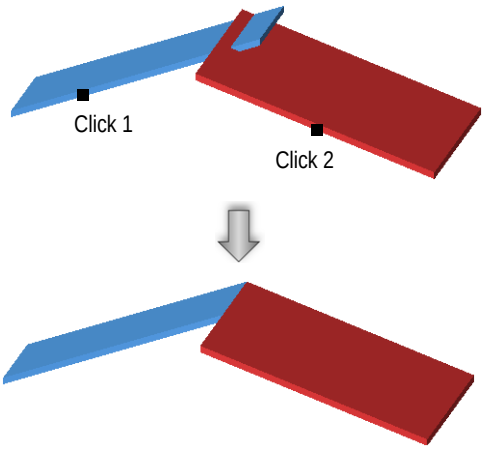


For intersecting faces, the first clicked face is trimmed to keep the clicked portion

Dual Object Connect mode



For non-intersecting faces, both faces are extended or trimmed to join

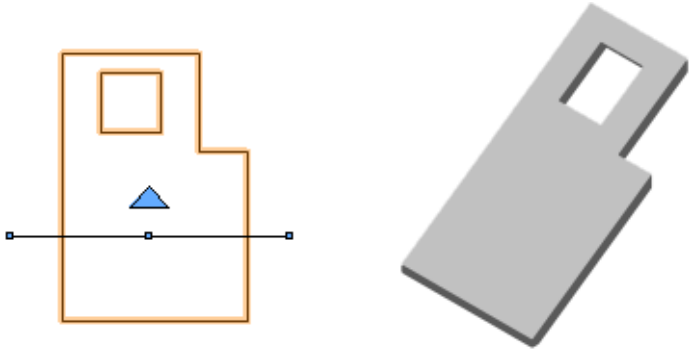


For intersecting faces, both faces are trimmed and the clicked parts of the faces are kept

With Use Vertical Thickness of Subject Roof Face enabled, the thickness of the boundary face (clicked second) changes to match the vertical thickness of the subject face (clicked first)

Creating Cutouts in a Roof Face

Use the **Clip Surface** command to add a cutout to a roof face. A cutout, for example, can be used to add a hole for a chimney that overlaps two or more faces of the roof.



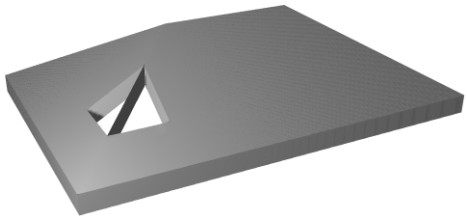
2D view 3D view

The edges of the cutout are mitered according to the selections made in the Create Roof Face dialog box during the creation of the roof face.

To add a cutout to a roof face:

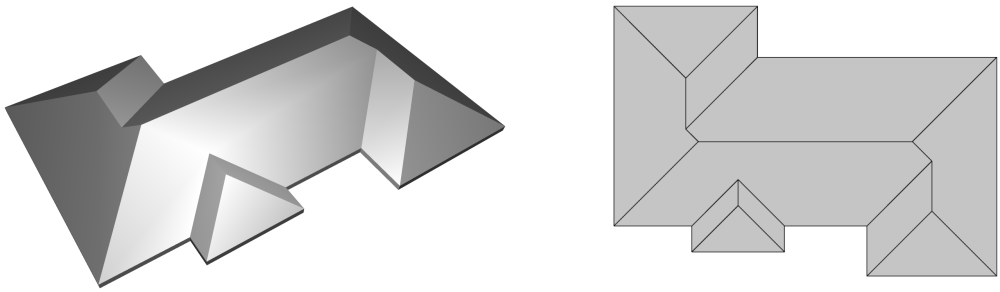
1. Select the roof face.
2. Create an object with the dimensions for the cutout and place it where the cutout will be located.
3. Select both objects.
4. Select **Modify > Clip Surface**.

   This creates the cutout in the roof face, leaving behind the object used to make the cutout.
5. Select the object used to make the cutout and delete it.



Adding Roof Elements to Roof Objects and Roof Faces
Creating Cutouts in a Roof Object

Creating Roof Objects



3D view 2D view

To create a roof object:

1. Select the walls, polygon, or polyline that the roof will be based upon.

Shift-click to select multiple walls. If the building has an attachment, such as a garage, that will have a different roof structure, then select only the desired walls.

2. Select the **Create Roof** command from the appropriate menu.

   - Fundamentals workspace: **Model > AEC > Create Roof**
   - Architect workspace: **AEC** > **Create Roof**
   - Landmark workspace: **Landmark > Architectural > Create Roof**
   - Spotlight workspace: **Spotlight > Architectural > Create Roof**

   The Create Roof dialog box opens. Enter the desired criteria to create the roof object. The parameters are illustrated in the roof diagram. Certain default parameters are suggested based on the selected roof parameters.

Click to show/hide the parameters.

Parameter	Description
Eave Profile	Select the roof edge appearance • Square: angled fascia, soffit edges • Vertical: vertical fascia, angled soffit edges • Horizontal: no fascia, horizontal soffit edges • Double: vertical fascia, horizontal soffit edges
(1) Vertical	Specifies the vertical length for a double miter eave
(2) Horizontal	Specifies the horizontal length for a double miter eave
(3) Thickness	Specifies the roof thickness
(4) Bearing Inset	Specifies how far the bearing wall cuts into the roof
(5) Roof Pitch	Specifies the roof pitch as an angle or rise:run ratio; click **Calculate** for an automatic calculation based on bearing height, eave height, and eave overhang
(6) Bearing Height	Specifies the height of the rafter plate or top plate above the wall layer elevation where the roof will be supported; click **Calculate** for an automatic calculation based on roof pitch, eave height, and eave overhang
(7) Eave Height	Specifies the height of the lowest portion of the roof; click **Calculate** for an automatic calculation based on roof pitch, bearing height, and eave overhang
(8) Eave Overhang	Specifies the distance that the roof extends beyond the bearing wall; click **Calculate** for an automatic calculation based on roof pitch, bearing height, and eave height
Layer	Assigns the roof object to a layer. By default, the roof is created in the active design layer. To create the roof in a different layer, select an existing layer from the list, or select **New Layer** to create a new layer.
Class	Assigns the roof object to a class. The standard roof class, Roof-Main, is available as an option and if it is selected, it is created in the file if it did not already exist. Alternatively, select **Class** to create a new class.
Create Accessories (Vectorworks Architect required)	Adds an attic, soffit, or fascia to the roof object; see “Adding Roof Accessories” on page 579
Retain Original Objects	Retains the source object(s) that formed the basis of the roof

3. Click **OK**.

   A hip roof is created over the selected object(s) using the criteria set in the Create Roof dialog box. The roof can be changed to a gable or Dutch hip shape; see “Changing the Roof Edge Shape” on page 573.

Editing Roof Objects
Applying Roof Textures
Roofs

Editing Roof Objects

Once a roof object has been created, there are multiple ways to edit it:

- Change roof edge shapes (such as hip or gable) using the **Selection** tool and the Edit Roof Settings dialog box.
- Create cutouts in the roof using the **Clip** tool or **Clip Surface** command.
- Reshape the roof object using the **Reshape** tool.
- Add an accessory: attic, soffit, or fascia (Vectorworks Architect required)

Additionally, the roof object parameters can be edited from the Object Info palette. Wall settings on the Object Info palette apply to all gable or Dutch hip walls in the roof; to adjust each gable wall individually, use the Edit Roof Settings dialog box.

Click to show/hide the parameters.

Parameter	Description
Bearing Inset	Specifies how far the bearing wall cuts into the roof
Thickness	Specifies the roof thickness (normally, this is the perpendicular thickness; when a roof has different slopes, the vertical thickness can be specified)
Applies To	When a roof has different slopes, the vertical thickness can be specified to avoid incorrect roof intersections. Select **Vertical Thickness** and enter the **Thickness** value
Eaves	Specify the eave type
Vertical	For Double Miter types, enter the vertical length of the eave
Gable Walls	An alert displays if you change these settings; confirm that you want to override the original settings
Show Walls	When gable walls exist, specifies whether the gable wall shows or is invisible
Thickness	When gable walls exist, specifies the gable wall thickness
Dutch Hip Walls	An alert displays if you change these settings; confirm that you want to override the original settings
Show Walls	When Dutch hip walls exist, specifies whether the Dutch hip wall shows or is invisible
Thickness	When Dutch hip walls exist, specifies the Dutch hip wall thickness

Changing the Roof Edge Shape
Reshaping Roof Objects
Adding Roof Elements to Roof Objects and Roof Faces
Creating Cutouts in a Roof Object
Adding Roof Accessories
Applying Roof Textures

Changing the Roof Edge Shape

By default, a hip roof is created with eave edges all around. Specific roof edges can be changed to a gable or Dutch hip shape.

To change the roof edge shape:

1. Select the roof with the **Selection** tool.

   Selection handles display for the roof.

2. Click on the selection handle of the edge to be edited.

   The Edit Roof Settings dialog box opens.

3. Change the roof parameters and select **Entire Roof** to apply the edits to all sections of the roof. If **Entire Roof** is not selected, the edits apply to the edge of the roof that was clicked.

Click to show/hide the parameters.

Parameter	Description
Roof Edge Shape	Select the basic shape of this roof edge: Eave, Gable, or Dutch Hip. The default is a hip roof, with an “Eave” edge on all sides. When this setting is changed, the available parameters and roof diagram reflect the selected roof edge shape.
Wall Thickness (Gable and Dutch Hip only)	Specifies the thickness of the wall created for gable and Dutch hip roof edges
Show Wall (Gable and Dutch Hip only)	Select this option to show the wall created for gable and Dutch hip roof edges; when deselected, the wall is invisible
Gable Overhang (Dutch Hip only)	Specifies how the gable wall on top of the Dutch hip face will be cut; enter zero for a flat gable wall, or enter the number of inches the gable wall will be inset from the gable roof edge
Gable Inset (Dutch Hip only)	Specifies how far the gable wall on top of the Dutch hip face will be from the edge of the hip face
Pitch (Eave and Dutch Hip only)	Specifies the roof pitch as an angle or rise:run ratio; click the calculator for an automatic calculation based on bearing height, eave height, and eave overhang
Bearing Height	Specifies the height of the rafter plate or top plate above the wall layer elevation where the roof will be supported; click the calculator for an automatic calculation based on pitch, eave height, and eave overhang
Eave Height	Specifies the height of the bottom-most portion of the roof; click the calculator for an automatic calculation based on pitch, bearing height, and eave overhang
Eave Overhang	Specifies the distance that the roof extends beyond the bearing wall, click the calculator for an automatic calculation based on pitch, bearing height, and eave height
Entire Roof (Eave and Dutch Hip only)	Select this option to apply the edited roof parameters to all available roof sections

4. Click **OK** to change the roof edge settings.

Changing a Hip Roof to a Gable Roof

To change a hip roof into a gable roof:

1. Select the roof with the **Selection** tool.

   Selection handles display for the roof.

2. Click on the selection handle of the face to change into a gable wall.

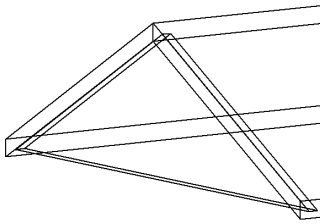
   The Edit Roof Settings dialog box opens.

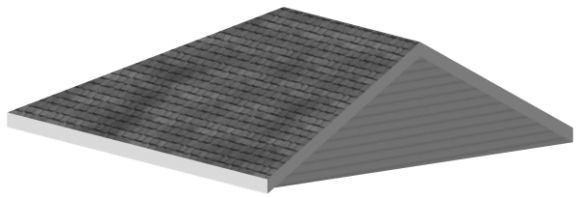
3. Select the **Gable** option.

   A diagram and parameters for a gable edge display.

4. Change the parameters as described in “Changing the Roof Edge Shape” on page 573, and click **OK**.

   A triangular wall is placed when the gable roof is created.





Changing a Hip Roof to a Dutch Hip Roof

To change a hip roof into a Dutch hip roof:

1. Select the roof with the **Selection** tool.

   Selection handles display for the roof.

2. Click on the selection handle of the face to change into a Dutch hip face.

   The Edit Roof Settings dialog box opens.

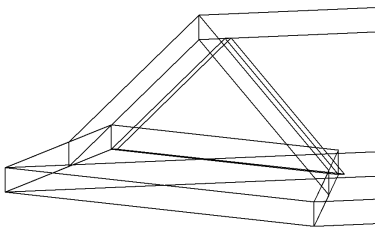
3. Select the **Dutch Hip** option.

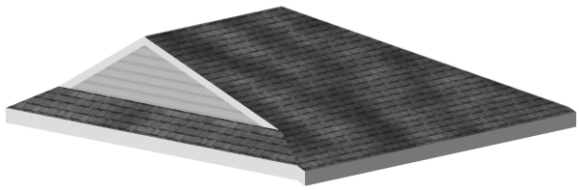
   A diagram and parameters for a Dutch hip edge display.

4. Specify a **Gable Inset** value to create the Dutch hip face, and change other settings as described in “Changing the Roof Edge Shape” on page 573.

5. Click **OK**.

   A triangular wall is placed when the Dutch hip roof is created.





Editing Roof Objects
Creating Cutouts in a Roof Object
Reshaping Roof Objects
Adding Roof Elements to Roof Objects and Roof Faces

Creating Cutouts in a Roof Object

You can use both the **Clip Surface** command and the **Clip** tool to clip holes in a roof object.

To create a cutout with the **Clip Surface** command:

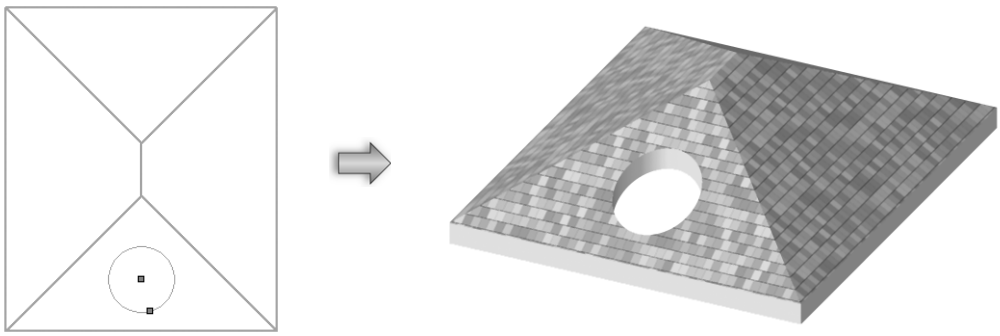
1. Draw one or more 2D cutout shapes (rectangle, circle, oval, polygon, polyline, or arc) on the roof object.

2. Select the cutout object(s) and the roof object.
3. Select **Modify > Clip Surface**.
4. The shapes are cut out of the roof, and the clip objects are selected.
5. Delete the clip objects.

To create a cutout with the **Clip** tool:

1. Select the roof object.
2. Click the **Clip** tool from the Basic palette.
3. From the Tool bar, select the Exclusion mode, and then select the appropriate clipping shape (rectangle, polygon, or circle).
4. Click and drag to create a marquee box.

   The roof is clipped as defined by the clipping shape.



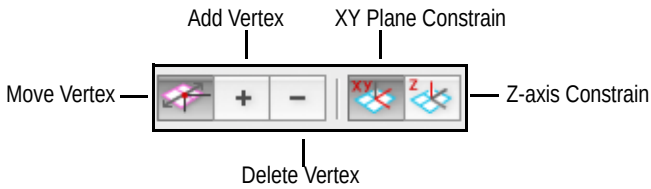
Editing Roof Objects
Creating Cutouts in a Roof Face

Reshaping Roof Objects

A roof object can be reshaped in a 2D or 3D view with the **Reshape** tool. The roof edge, roof ridge, and roof slope can be changed.

> In Top/Plan view, the **Reshape** tool can be used for basic roof shape editing; see "2D Reshape Modes" on page 1044. However, the **Reshape** tool should be used in a 3D view to edit the roof edge and roof ridge location.

Five modes are available when both the **Reshape** tool and a roof object is selected in a 3D view.



Mode	Description
Move Vertex	Adjusts the position of a selected roof vertex, constrained horizontally or vertically in combination with the Z-axis Constrain and XY Plane Constrain modes
Add Vertex	Adds a peak (vertex) to a roof
Delete Vertex	Deletes a roof peak (vertex)

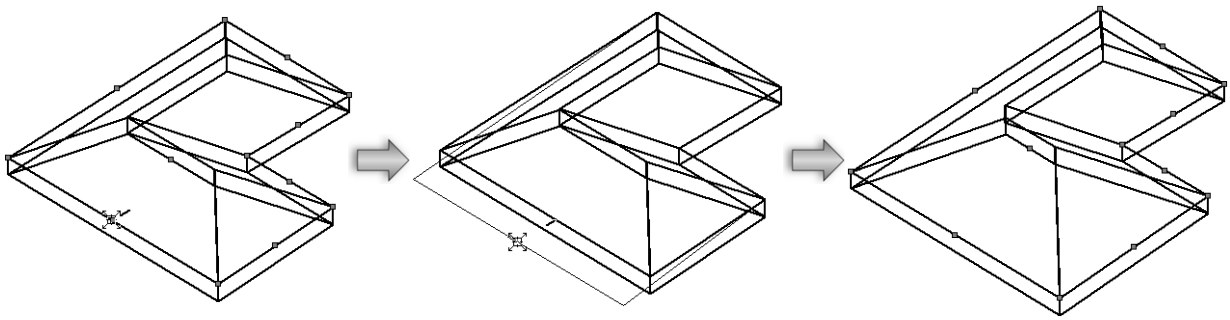
Mode	Description
XY Plane Constrain	In 3D Reshape mode, reshapes the roof object horizontally, constrained along the X and/or Y axes
Z-axis Constrain	In 3D Reshape mode, reshapes the roof object along the Z axis to change the height of a roof ridge or eave

Reshaping the Roof Along the X and Y Axis

To reshape the roof horizontally:

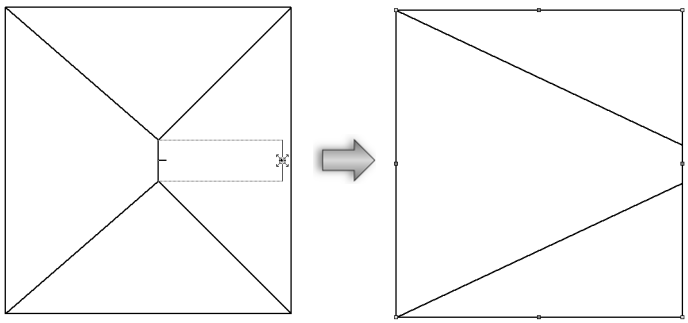
1. In a 3D view, select the roof object.
2. Click the **Reshape** tool from the Basic palette, and select the **XY Plane Constrain** mode.
3. Position the cursor over a roof handle.

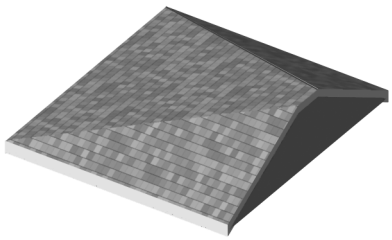
   When the cursor is over a handle, the standard arrow cursor changes into an unfilled, four-way arrow.
4. Click-drag the handle to change the roof handle location, or enter specific X and Y distance values in the Data bar.



The roof edge, ridge, or eave handle location can be changed. The handle movement is constrained along the X or Y axis (the active layer plane); the height of the roof element cannot be changed in this mode.

Change the slope of a roof face by moving the ridge handle. If the reshape results in a roof slope that approaches or exceeds 90 degrees, you are prompted to create a gable wall if desired.





Rendered View

5. Click when the handle is at the desired location.

Reshaping the Roof Along the Z Axis

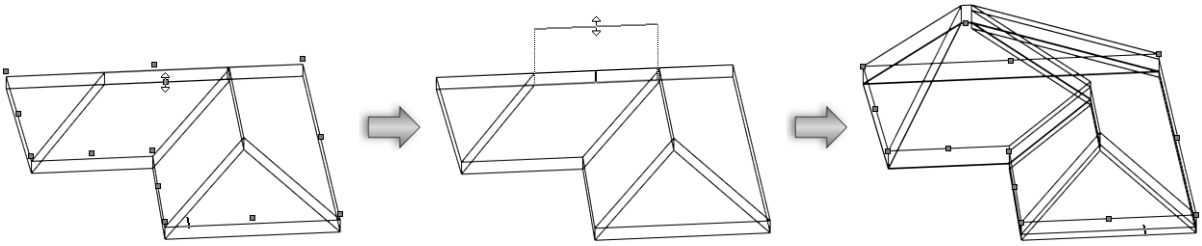
To change the height of a roof ridge or eave:

1. In a 3D view, select the roof object.
2. Click the **Reshape** tool from the Basic palette, and select the **Z-axis Constrain** mode.

3. Position the cursor over a roof ridge or eave handle.

   When the cursor is over a handle, the standard arrow cursor changes into two-way hollow arrow.

4. Click-drag the handle to change the roof handle location, or enter the specific Z axis distance in the Data bar.



   The roof ridge or eave handle location can be changed. The handle movement is constrained along the Z axis (vertically); only the height of the roof element can be changed in this mode.

5. Click when the handle is at the desired location.

Adding a Roof Vertex

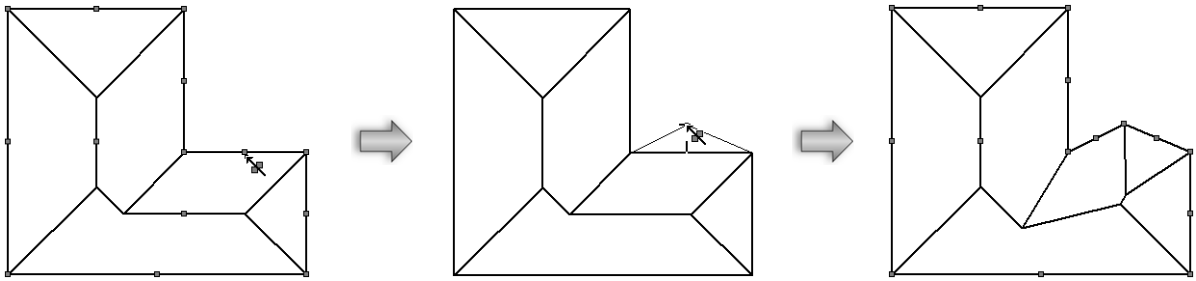
To add a vertex to a roof:

1. In a 3D view, select the roof for adding the vertex.
2. Click the **Reshape** tool from the Basic palette, and select **Add Vertex** mode.
3. Position the cursor over one of the edge or eave handles.

   The standard arrow cursor changes into a single-headed, filled arrow with shaded boxes on either side of the shaft.

4. Click-drag the mouse to add a vertex to the roof edge or eave.
5. Click when the vertex is at the desired location.

   The roof is automatically reshaped to accommodate the new vertex.



Deleting a Roof Vertex

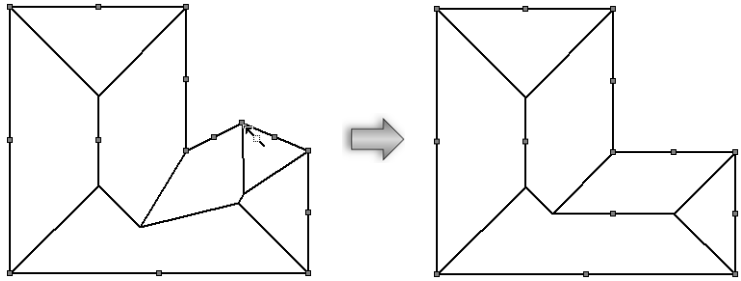
To delete a vertex from a roof:

1. In a 3D view, select the roof with the vertex to be deleted.
2. Click the **Reshape** tool from the Basic palette, and select **Delete Vertex** mode.
3. Position the cursor over the eave vertex to delete.

   The standard arrow cursor changes into a single-headed, filled arrow with a hollow diamond in the shaft.

4. Click the vertex.

   The vertex is removed and the roof is reshaped to the remaining vertices.



Reshaping Objects
Editing Roof Objects

Adding Roof Accessories

In the Vectorworks Architect product, attic, soffit, and fascia roof accessories can be inserted when a roof object is being created, or they can be added, modified, or deleted later on. If the roof is reshaped, the accessories are automatically regenerated to suit the new shape.

The accessories are part of the roof object. To modify or remove them later on, select the roof object, and change the accessory settings from the Object Info palette.

Roof accessories cannot be added to roof face objects. See “Creating Roof Objects” on page 571.

To add roof accessories:

1. With the appropriate walls selected, select **AEC** > **Create Roof**.

   The Create Roof dialog box opens.

2. Click **Create Accessories**.

   The Create Roof Accessories dialog box opens.

Click to show/hide the parameters.

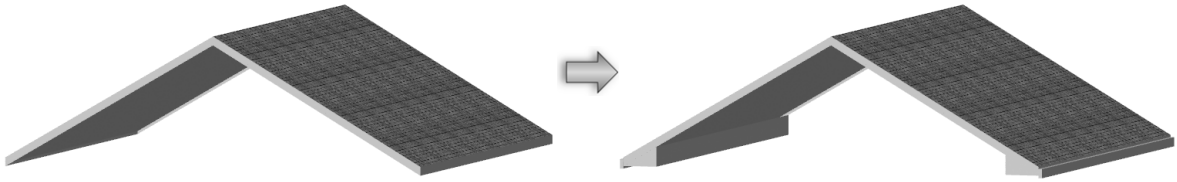
Parameter	Description
Insert Attic	Select to add an attic to the roof
Attic Settings	Opens the Edit Attic Settings dialog box for specification of attic parameters
Insert Soffit	Select to add a soffit to the roof
Soffit Settings	Opens the Edit Soffit Settings dialog box for specification of soffit parameters
Insert Fascia	Select to add fascia to the roof
Fascia Settings	Opens the Edit Fascia Settings dialog box for specification of fascia parameters

3. Select the accessories to be added to the roof object.
4. For each selected accessory, click the corresponding settings button.

   A dialog box displays with parameters for that accessory.

5. Enter accessory parameters as described in the following sections and click **OK** to return to the Create Roof Accessories dialog box.
6. Repeat steps 4 and 5 until all desired accessory parameters have been entered.

7. From the Create Roof Accessories dialog box, click **OK** to return to the Create Roof dialog box; click **OK** again to create the roof object with the selected accessories.



Roof with no accessories

Roof with soffit and fascia added

Creating Roof Objects
Inserting an Attic
Inserting a Soffit
Inserting Fascia

Inserting an Attic

To add an attic accessory to a roof:

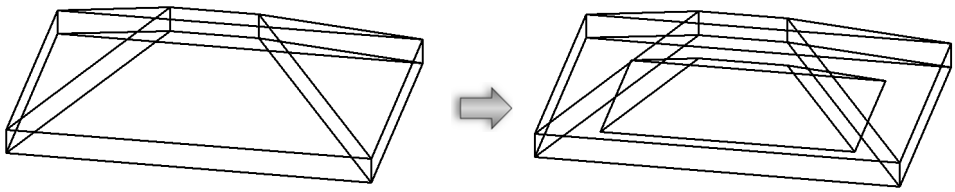
1. From the Create Roof Accessories dialog box, select **Insert Attic**, and click **Attic Settings**.

   The Edit Attic Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Full Height	Specifies that the attic space is the full height of the attic area
Partial Height	Specifies that the attic space is the height entered in the box provided

2. Enter the attic parameters and click **OK** to return to the Create Roof Accessories dialog box.



Adding Roof Accessories

Inserting a Soffit

The soffit accessory has a “pork chop” profile.

To add a soffit accessory to a roof:

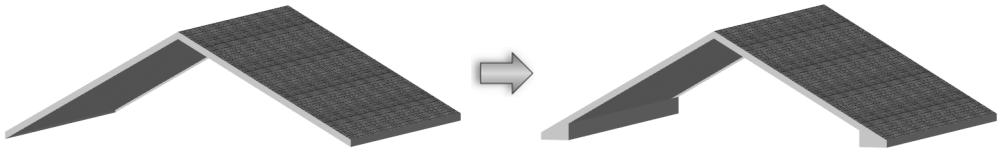
1. From the Create Roof Accessories dialog box, select **Insert Soffit**, and click **Soffit Settings**.

   The Edit Soffit Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Recess	Specifies the distance between the bottom edges of the roof rafters and the soffit
Trim Depth	Specifies the distance between the outer edges of the rafters and inside edge of the soffit

2. Enter the soffit parameters and click **OK** to return to the Create Roof Accessories dialog box.



Adding Roof Accessories

Inserting Fascia

The fascia accessory has a rectangular profile.

To add a fascia accessory to a roof:

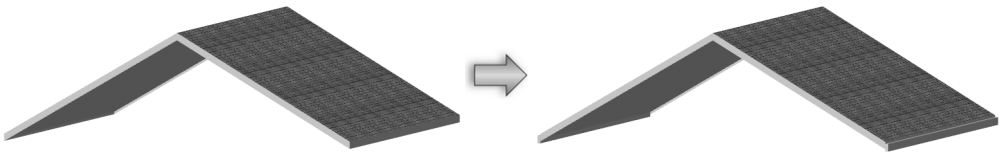
1. From the Create Roof Accessories dialog box, select **Insert Fascia**, and click **Fascia Settings**.

   The Edit Fascia Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Fascia Width	Specify a width for the fascia accessory
Fascia Height	Specify a height for the fascia accessory

2. Enter the fascia parameters and click **OK** to return to the Create Roof Accessories dialog box.



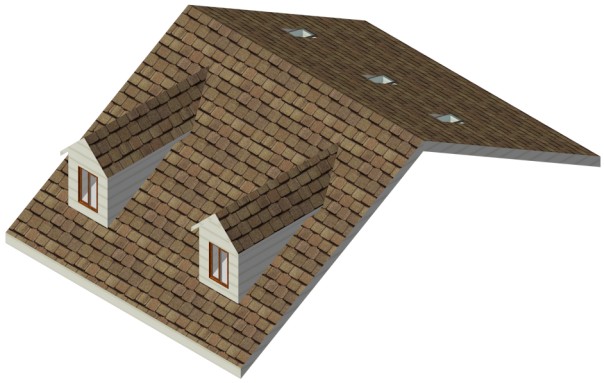
Adding Roof Accessories

Adding Roof Elements to Roof Objects and Roof Faces

Once a roof object or a roof face has been created, roof elements, such as dormer windows and skylights, can be added.

Creating Dormer Windows
Creating Skylights

Creating Dormer Windows



You can create a wide variety of dormer windows in roof objects and roof faces. There are five styles to select from: trapezium, gable, shed, hip, and bat; each gable type has unique parameters. The editing parameters available in the Edit Roof Element dialog box depend on the style of dormer chosen.

Dormer walls are always drawn in a clockwise direction for easy texture application.

To create a gable dormer window in a roof object or roof face:

1. Select **View > Standard Views > Top/Plan**.
2. Select **Window > Palettes > Resource Browser**.
3. Select a window symbol.

   The window must be a symbol, not a plug-in object.
4. Select **Resources > Make Active**. The **Symbol Insertion** tool is automatically activated from the Basic palette.
5. Click to place the symbol in the roof object or roof face.

   The Edit Roof Element dialog box opens.
6. Click **Edit Dormer**.
7. Select the dormer style.

   The parameters automatically change according to the selected dormer style, with values for placing the dormer at the location specified with the mouse click.

Click to show/hide the parameters.

Parameter	Dormer Style	Description
Center vertically	All	Places the center of the window symbol in the center of the available vertical space in the front face of the dormer; the normal insertion point is not used
Offset from top	All	Locates the top of the window symbol a set distance from the top of the dormer face; the normal insertion point is not used
Height offset	All	Indicates the vertical distance from the top of the point of engagement with the roof, or where the roof and the dormer meet, to the bearing point, which is usually along the top of the bearing wall
Building line offset	All	Specifies the distance from the building outline to the plan center of the window symbol

Parameter	Dormer Style	Description
Offset from corner	All	Sets the distance from the corner of the roof to the center of the dormer; the roof corner that the measurement is taken from is always to the left of the dormer when facing it
Top width	Trapezium, Bat	Determines the width of the top roof and sets the front face's trapezoid shape; the front face is always symmetrical when using this option
Right slope	Trapezium, Gable, Hip	Determines the angle of the right edge of the front face; the front face can be asymmetrical when using this option
Left slope	Trapezium, Gable, Hip	Specifies the angle of the left edge of the front face. Along with the **Right Slope**, this dimension determines the top width of the front face. **Right Slope** must be selected for this option to be available.
Bottom width	Trapezium, Gable, Bat	Sets the width of the bottom edge of the front face; works in conjunction with either the **Top Width** or the **Left** and **Right Slope** entries and is required
Slope	Trapezium, Shed, Bat	Indicates the angle of the pitch of the top dormer roof as measured from a horizontal line
Width	Gable, Shed, Hip	Horizontal distance of the front face of the dormer
Front Slope	Hip	Indicates the angle of the pitch of the front face of the dormer roof as measured from a vertical line
Height	All	Specifies the elevation height of the front face of the dormer; determines the plan depth of the dormer
Top width	Bat	Indicates the width of the top of the roof as measured along the front face of the dormer
Bottom Height	Bat	Distance from the bottom of the dormer to the beginning of the compound curves of the roof as measured along the front face of the dormer
Depth	All	Sets the plan distance from the point of engagement with the roof to the front face of the dormer; determines the elevation height of the front face of the dormer
Overhang	Gable, Shed, Hip	Amount of roof extension past the dormer's front roof face
Control Point	Bat	Point where the two curves of the roof meet. This option controls the location of that point from the side edge of the dormer as measured along the roof. The location of this point determines the depth of the curves that make up the roofline.

8. Click **OK**.

   The gable dormer with window is created and placed according to the parameters specified. A hole is automatically created in the roof where the dormer walls exist.

   When a gable dormer is created on a roof face, the Object Info palette of a selected roof face object contains two additional **Dormer Settings** parameters, Wall Thickness and Roof Thickness. These thickness values apply to the dormer(s) in the roof face.

   Click here for a video tip about this topic (Internet access required).

Editing Dormer Windows

Editing Dormer Windows

Once created, the dormer window parameters can be edited.

To edit a dormer window:

1. Select the dormer.

   Selection handles display around the roof or roof face and at the location of each dormer.

2. Position the cursor over the selection handle for the dormer and click.

   The Edit Roof Element dialog box opens.

3. Select **Edit Dormer** and change the desired parameters as described in “Creating Dormer Windows” on page 582.

   To remove the dormer completely, select **Remove Object**.

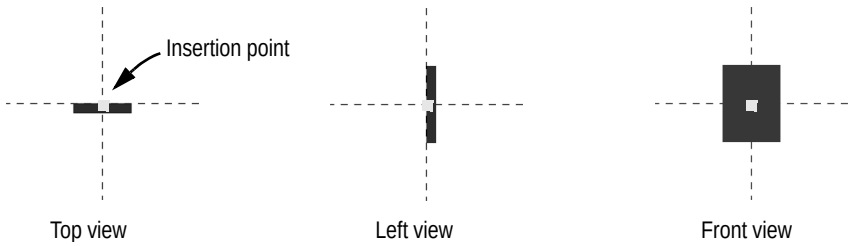
4. Click **OK**.

Creating Skylights

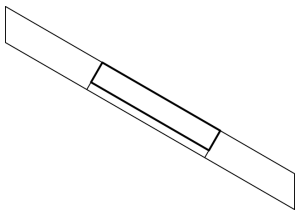
A full skylight, complete with a window symbol, can be placed in the roof object or roof face.

A 3D-only window symbol is required for creating a skylight. An existing hybrid symbol can be converted to a 3D symbol; place the symbol in the drawing and set its parameters, and then switch to a 3D view. Select the symbol, and then select **Modify Convert > Convert to Group**. Select the **Convert all sub-objects** option in the Convert to Group Options dialog box. With the group selected, choose **Modify > Create Symbol** to create a 3D symbol from the group, and select the **On Edge** option for **Insert in Walls**.

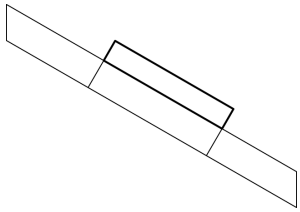
When creating your own skylight symbols, set the insertion point of the symbol at the back and center of the symbol.



The insertion point of the symbol determines whether a skylight will be flush or surface-mounted.



Square miter flush-mounted



Square miter surface-mounted

Inserting a Skylight

To insert a skylight:

1. Select **Window > Palettes > Resource Browser**.
2. Select a 3D symbol to use in the skylight. Hybrid and 2D window symbols will not work for skylights, though hybrid symbols can be used to create a cutout.
3. Select **Resources > Make Active**, or double-click on the symbol. The **Symbol Insertion** tool is automatically activated from the Basic palette.
4. Switch to Top/Plan view.
5. Click to place the symbol in the desired location in the roof object or roof face.

   The Edit Roof Element dialog box opens.
6. Click **Edit Skylight**.

   The skylight parameters are displayed.

Click to show/hide the parameters.

Parameter	Description
Offset from corner	Specifies the distance from the edge of the roof to the center of the skylight symbol
Offset from building line	Sets the distance from the edge of the building to the center of the skylight symbol
Do not insert symbol	Select to create a cutout in the roof without inserting the window symbol
Remove object	Deletes the skylight from the roof
Edit dormer	Accesses the dormer parameters instead of the skylight parameters

7. Click **OK** to create the skylight (or cutout).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Skylights

Editing Skylights

To edit a skylight:

1. Select the skylight.

   A selection handle displays at the skylight location.

2. Position the cursor over the selection handle for the skylight and click.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Edit Roof Element dialog box opens, with the skylight parameters displayed.

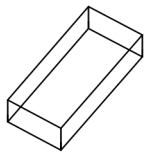
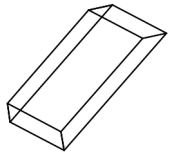
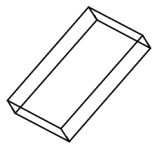
3. Edit the parameters as described in “Inserting a Skylight” on page 585.

   To remove the skylight completely, select **Remove object**.

4. Click **OK**.

   The drawing area displays the specified changes for the selected skylight.

The skylight miter parameters can be changed from the Object Info palette.

Skylight Miter	Description
Vertical	Cuts the roof vertically at both the top and bottom of the skylight 
Splayed	Cuts the roof horizontally at the top of the skylight, and vertically at the bottom 
Square Cut	Cuts the roof perpendicular to the roof at both the top and bottom of the skylight 

Windows

In the Vectorworks Design Series products, the window object is inserted with the **Window** tool. In the Vectorworks Fundamentals product, the window object is inserted from an object library through the Resource Browser. All Vectorworks products include a window object containing parameters similar to those described here. The Vectorworks Architect and Landmark products include enhanced capabilities.

The Vectorworks Architect and Landmark products also include window symbols in standard sizes and various configurations, ready to install in walls. These products also include the ability to insert windows into curtain walls as special "curtain wall windows." Additionally, the Vectorworks Architect product provides extensive libraries of window symbols, listed by configuration and size.

> The **Update Plug-in Objects** command may need to be run on files containing windows that were created in an earlier version of the Vectorworks Architect or Landmark product. This command converts the windows to the latest format; see "Migrating from Previous Versions" on page 28.

A window can be customized by adjusting various parameters, and then saved as a symbol so that its parameters are preset upon insertion. This eliminates the need to repeatedly apply parameters, maximizes memory efficiency, and allows global editing of symbols. See "Creating New Symbols" on page 239.

If a custom window is unique (that is, it has only one instance) in a project, creating a symbol from it is unnecessary. However, making a symbol of the installed window makes it easy to reuse it in other drawings should there be a need to do so in the future.

After the symbol is created, it can be selected from the Window Settings dialog box (by selecting **Use Symbol Geometry** as described in "Window Settings: General Pane" on page 595), where it can take advantage of functionality such as embedded IDs and data, cavity wrapping, splays, and wall offsets. To enable automatic ID labeling, from the ID Tag pane of the settings dialog box, select **Include on Schedule**, and specify the ID parameters. If necessary, set the class to which the ID is assigned to visible. Alternatively, set the automatic ID labeling through the **ID Label** tool; see "Using the ID Label Tool" on page 1262.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Windows in Vectorworks Fundamentals

To insert a window in Vectorworks Fundamentals:

1. Insert a window symbol from the Resource Browser (see "Inserting Symbols" on page 242). The direction of a window inserted in a wall can be changed later by clicking **Flip** on the Object Info palette or from the context menu.

   A library with an editable window symbol is included with the Vectorworks Fundamentals product. If this content was not downloaded during installation, download it at any time by selecting **Help > Download Content** (see "Resource Libraries" on page 219).

   The **Window** tool is available for Vectorworks Fundamentals software but is not present in the Fundamentals workspace. It can be added to the Fundamentals workspace (see "Creating or Editing Workspaces" on page 1835) and subsequently used to insert window objects.

2. Click the **Settings** button on the Object Info palette to open the Window Settings dialog box. Select each pane and specify the window parameters. Click **OK** to set the window parameters and close the dialog box.

   Several features of the window are described as "interior" or "exterior." These include trim, shutters, and wall-wrap parts. These elements are determined based on the internal and external faces of the wall, not on the window's exterior direction. The left side of the wall (as viewed along the wall direction) is always "exterior," and the right side is "interior" (see "Wall Direction" on page 506). Flipping the window does not flip these elements.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Window Settings: Preview

Click to show/hide the parameters.

Parameter	Description
3D Preview	Dynamically displays a 3D preview of the window with the currently selected parameters
View	Select the standard view for the 3D preview of the window
Render	Select the render mode for the 3D preview of the window
Top/Plan Preview	Dynamically displays a Top/Plan preview of the window with the currently selected parameters

Window Settings: General Pane

Click to show/hide the parameters.

Parameter	Description
Top Shape	Select the shape of the top of the window
Transom	Select whether to add a transom above the window; transom parameters are set from the Transom pane
Rise	When applicable for the selected **Top Shape**, enter the distance between the start of the top shape to the top of the window
Spring	Enter the distance traveled above the bottom pane before the top shape starts; if Square is selected as the **Top Shape**, the spring is the total height
Sash	Select the window type. Depending on the selection, a variety of window settings are enabled or disabled automatically, as described in the following sections.
Use Symbol Geometry	Select **Use Symbol Geometry** and click **Browse** to select a window symbol from the current file's resources. Select a symbol from the graphical list of **Symbols** and click **OK**. The symbol **Name** displays in the Window Settings dialog box and the preview is updated with the selected symbol. When a symbol is selected, all fields pertaining to the window's geometry are disregarded; however, parametric values are still available for scheduling and reporting purposes only.
Size Reference: Unit Size	For a unit size window, the width/height of the window is measured at the outside of the jamb; it includes the transom and sash. Set the **Width** and **Height**.
Elevation in Wall	Enter the window elevation
Elevation Reference	Select either the sill or the head of the window as the elevation reference point
Plan Wall Offset	Enter the distance to offset the window insertion point from the centerline of the wall

Window Settings: 2D Visualization Pane

Click to show/hide the parameters.

Parameter	Description
Show Plan Detail	Select to display all the window details in Top/Plan view

Parameter	Description
Draw Wall Lines	Select whether to draw wall lines at all times; wall lines are drawn in the Ceiling-Main class to easily create reflected ceiling plans
Set Attributes By	Select whether to set the 2D graphic attributes and visibility of the listed window parts by Object, Line Style, or Class. • Object: disables line style and weight controls for individual parts, and uses the line style and weight settings for the window object. • Line Style: controls line style and weight for the individual parts; these settings override the window object's settings. • Class: sets the line style and weight of individual parts by class. Select a class for each part to control its visibility and appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the window class to place the part in the same class as the window object.

Window Settings: 3D Visualization Pane

3D Visualization options are not enabled for the Opening **Sash** configuration.

Click to show/hide the parameters.

Parameter	Description
3D Hinge Direction Marker	
Show Interior/ Exterior Marker	Select to draw lines representing the hinge direction of the window sashes on the interior/ exterior side of the window in 3D
Use Hinge Marker Class Attributes	Select to set the hinge direction marker's attributes by class
Hinge Direction Marker Class	Select a class to control visibility and, if **Use Hinge Marker Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default window class to place the part in the same class as the window object.
Marker Points Towards	Set the hinge direction marker to point toward the hinge or toward the handle
Show 3D Open	Select to draw 3D window sashes as open at the specified **Open Angle**
Clerestory	Enabling **Clerestory** draws the window as dashed if the sill is higher than the specified elevation and draws the window as non-breaking (solid) if the sill is below the specified **Dash Height**

Window Settings: ID Tag Pane

Click to show/hide the parameters.

Parameter	Description
Include on Schedule	Select whether to include this window's information in the window schedule and also set the window ID tag to visible
ID Prefix	Assigns alphanumeric information before the numerical label value; adding prefix information is optional

Parameter	Description
ID Label	Assigns a numerical value to the ID; this number increments automatically if the auto-increment option is chosen in the ID Settings dialog box (see “Using the ID Label Tool” on page 1262)
ID Suffix	Assigns alphanumeric information after the label value; adding suffix information is optional
ID Class	Select a class to control visibility and, if **Use ID Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default window class to place the part in the same class as the window object.
Use ID Class Attributes	Select to use the ID class attributes for the bubble line, leader line, and leader line marker
Keep ID Horizontal	When selected, automatically rotates the ID so that it is horizontal
Bubble Shape	Select the ID tag bubble shape
Bubble Size	Enter the minimum bubble size (this value represents the bubble size times the layer scale; the bubble shape is maintained relative to the text inside it for ID bubble uniformity throughout the drawing file)
Bubble Line Attributes	Select the bubble line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness
Show Leader	Select to draw a leader line from the ID tag to the window object
Line Attributes	Select the leader line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness
Include Marker	Select whether to include a marker for the ID leader line and choose the desired style from the marker list, or select **Custom** to create a custom marker. Select **Edit Marker List** to open the Edit Marker List dialog box; see “Editing the Marker List” on page 1103.

Window Settings: Jamb and Sash Pane

Click to show/hide the parameters.

Parameter	Description
Jamb	
Jamb Width	Enter the face width of the window jamb (parallel to the wall)
Jamb Depth	Enter the depth of the window jamb (perpendicular to the wall)
Use Wall Depth	Select to set the jamb depth to the overall wall depth
Sash	
Sash Width	Enter the sash width
Sash Depth	Enter the sash depth
Unequal Sash	To create a double hung window with an unequal sash height, select to customize the top sash height, and enter the desired scale factor for the sash. Select to customize the width of the left and right sliders for a picture window slider, and enter the desired scale factor for the sashes. The sash scale factor applies to both the left and right sliders.

Parameter	Description
Shim Gap	Enter the distance between the jamb exterior face and the rough opening (for the Opening **Sash** configuration, the **Shim Gap** is automatically set to 0)
Masonry Module	Enter the masonry module dimension

Window Settings: Sill Pane

Click to show/hide the parameters.

Parameter	Description
Include Sill	Select whether to include a sill
Sill Type: Timber	Specify the sill lip thickness, amount of keep for the sill, sill height, total sill depth, sill extension width, stool lip thickness, and stool nose dimension

Window Settings: Transom Pane

Click to show/hide the parameters.

Parameter	Description
Sash	
Sash Width	Enter the transom sash width
Sash Depth	Enter the transom sash depth
Mullion	
Mullion Width	Enter the transom mullion width
Mullion Depth	Enter the transom mullion depth
Muntins	
Pattern	Select the transom muntin pattern
Vertical/Horizontal Bars	For colonial muntins, enter the number of vertical/horizontal muntin bars Prairie and renaissance muntins come with pre-set configurations; therefore, any number of **Vertical/Horizontal Bars** specified are disregarded
Bar Depth/Width	Enter the transom muntin bar width/depth
Bar Offset	Enter the transom muntin bar offset
Number of Rays	For sunburst patterns, enter the number of muntin rays

Window Settings: Trim Pane

Click to show/hide the parameters.

Parameter	Description
Include Interior/ Exterior Trim	Select whether to include interior/exterior trim
Width/Depth	Enter the interior/exterior trim width and depth

Parameter	Description
Include Trim Under Stool/Sill	Select whether to include interior/exterior trim along the window bottom

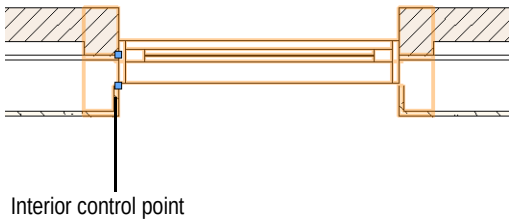
Window Settings: Muntins Pane

Click to show/hide the parameters.

Parameter	Description
Top Sash/Bottom Sash	Select which sashes should have muntins applied to them
Pattern	Select the muntin pattern
Vertical/Horizontal Bars	For colonial muntins, enter the number of vertical/horizontal muntin bars Prairie and renaissance muntins come with pre-set configurations; therefore, any number of **Vertical/Horizontal Bars** specified are disregarded
Bar Depth/Width	Enter the muntin bar width/depth
Bar Offset	Enter the muntin bar offset
Number of Rays	For sunburst patterns, enter the number of muntin rays

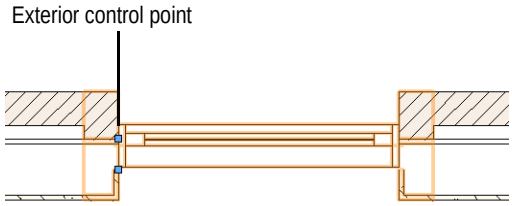
Window Settings: Interior Wall Detail Pane

Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of interior wall components to wrap; when interior wall components are specified, the window enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the interior wall; the splay can be specified either as an angle/diagonal or with width and depth values
(1) Splay Angle	Enter the interior wall splay angle
(2) Splay Diagonal	Enter the interior wall splay diagonal value
(3) Splay Width	Enter the interior wall splay width
(4) Splay Depth	Enter the interior wall splay depth

Window Settings: Exterior Wall Detail Pane

Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of exterior wall components to wrap; when exterior wall components are specified, the window enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the exterior wall; the splay can be specified either as an angle/diagonal or with width and depth values
(1) Splay Angle	Enter the exterior wall splay angle
(2) Splay Diagonal	Enter the exterior wall splay diagonal value
(3) Splay Width	Enter the exterior wall splay width
(4) Splay Depth	Enter the exterior wall splay depth

Window Settings: Classes Pane

The visibility of the overall 3D window is controlled by the **Class** setting on the Object Info palette; part settings are controlled from the Window Settings dialog box.

Click to show/hide the parameters.

Parameter	Description
Window parts	Select a class for each window part to control its fill color and/or texture and visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the window class to place the part in the same class as the window object.

Window Settings: Data Pane

Certain data fields represent calculated values and cannot be edited; as a result, the **Field Name** and **Field Value** appear dimmed for those data fields.

Click to show/hide the parameters.

Parameter	Description
Field Name	Select the data field from the list and its name displays beneath the list; information associated with the **Field Name** can be specified in the **Field Value** area. The user fields can include additional user-defined information with the window.
Field Value	Enter data for use in the window schedule

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Windows
Inserting Windows in Vectorworks Design Series
Window Properties
Creating Window Schedules
Creating a Custom Window Sash Opening
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Window Terminology Specific to the United Kingdom
Editing Symbols in Walls

Inserting Windows in Vectorworks Design Series

To insert a window:

1. Select the **Window** tool from the Building Shell tool set.

   Alternatively, if placing a curtain wall window into a curtain wall (Vectorworks Architect or Landmark required), select a panel with the **Edit Curtain Wall** tool, and then right-click (Windows) or Ctrl-click (Mac) on the panel and select **Insert Window** from the context menu. The window is automatically inserted as a curtain wall window.

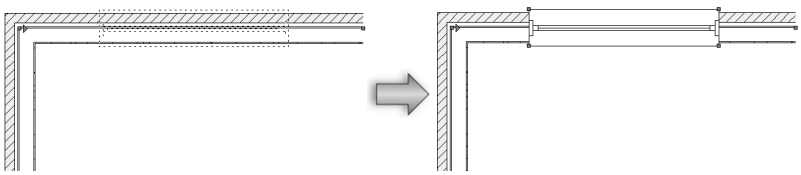
2. If this is the first time a window is inserted into the drawing, or if you wish to change the default preferences that apply to subsequently placed windows, click **Preferences** from the Tool bar.

   The Window Settings dialog box opens. The settings are grouped into several panes of related parameters, which are listed on the left side of the dialog box. Select each pane and specify the window parameters. Click **OK** to set the window parameters and close the dialog box. These parameters can be edited for placed windows from the Object Info palette, as described in “Window Properties” on page 606.

3. Click in the drawing area or in a wall (standard wall or curtain wall) to set the insertion point of the window, and click again to set the rotation. When inserting a window into a wall, place the second click on the exterior side of the wall to establish the exterior direction of the window. The direction of a window inserted in a wall can be changed later by clicking **Flip** on the Object Info palette.

   Several features of the window are described as “interior” or “exterior.” These include trim, shutters, and wall-wrap parts. These elements are determined based on the internal and external faces of the wall, not on the window’s exterior direction. The left side of the wall (as viewed along the wall direction) is always “exterior,” and the right side is “interior” (see “Wall Direction” on page 506). Flipping the window does not flip these elements.

   If you are inserting a corner window, place the window in a wall that is attached to another wall. Once parameters are set, the window moves to the nearest corner automatically. This represents one half of the corner window assembly.



Window Settings: Preview

Click to show/hide the parameters.

Parameter	Description
3D Preview	Dynamically displays a 3D preview of the window with the currently selected parameters
View	Select the standard view for the 3D preview of the window
Render	Select the render mode for the 3D preview of the window
Top/Plan Preview	Dynamically displays a Top/Plan preview of the window with the currently selected parameters

Window Settings: General Pane

Click to show/hide the parameters.

Parameter	Description
Window Shape	Select the window shape All window shapes other than rectangular are fixed glass; only the appropriate parameters on each pane of the Window Settings dialog box are available for these fixed glass shapes.
Top Shape	Select the shape of the top of the window
Transom	Select whether to add a transom above the window; transom parameters are set from the Transom pane
Rise	When applicable for the selected **Top Shape**, enter the distance between the start of the top shape to the top of the window
Spring	Enter the distance traveled above the bottom pane before the top shape starts; if Square is selected as the **Top Shape**, the spring is the total height
Sash	Select the window type. Depending on the selection, a variety of window settings are enabled or disabled automatically, as described in the following sections. For the Vectorworks Architect or Landmark product, to create a simple opening in a wall for defining a window break without creating any geometry to represent the window, select the Opening option. To define a custom window sash, select Custom **Sash** and click **Custom Sash Options** to open the Custom Sash Options dialog box.
Use Symbol Geometry	Select **Use Symbol Geometry** and click **Browse** to select a window symbol from the current file's resources. Select a symbol from the graphical list of **Symbols** and click **OK**. The symbol **Name** displays in the Window Settings dialog box and the preview is updated with the selected symbol. When a symbol is selected, all fields pertaining to the window's geometry are disregarded; however, parametric values are still available for scheduling and reporting purposes only.
Size Reference	Select the appropriate option to define the window width and height; the **Size Reference** selection automatically updates the **Width** and **Height** fields, but does not change the size of the window: • Sash Opening - this represents the window opening including the sash and glass, but excluding the transom and jamb; this is typically used to show the window opening for emergency exit purposes • Unit Size - this represents the current overall window width and height, including the transom and sash; this is the default setting for imperial units • Rough Opening - this represents the current overall window width and height, including the transom, sash, and shim gap (for the Opening **Sash** configuration, the **Size Reference** is automatically changed to Rough Opening); this is the default setting for metric units Vectorworks Architect or Landmark product required for Unit Size and Rough Opening.

Parameter	Description
Width/Height	The width/height of the window is measured based on the **Size Reference** selection: • Sash Opening - width/height is measured at the outside of the sash • Unit Size - width/height is measured at the outside of the jamb • Rough Opening - width/height measures the rough opening For circle, half circle, hexagon, and octagon shaped windows, the height parameter is disabled, and the parameter is automatically calculated based on the window's width.
Elevation in Wall	Enter the window elevation
Elevation Reference	Select either the sill or the head of the window as the elevation reference point
Plan Wall Offset	Enter the distance to offset the window insertion point from the centerline of the wall
Curtain Wall Window (Vectorworks Architect or Landmark required)	Indicates that the window will be placed in a curtain wall, where its width and height are based on the surrounding panel size and the bottom frame where it is placed. A curtain wall window does not have as many available parameter options as a standard window. Configurations are limited to those that apply to curtain wall placement.

Window Settings: 2D Visualization Pane

Click to show/hide the parameters.

Parameter	Description
Show Plan Detail	Select to display all the window details in Top/Plan view
Draw Wall Lines	Select whether to draw wall lines at all times; wall lines are drawn in the Ceiling-Main class to easily create reflected ceiling plans
Set Attributes By	Select whether to set the 2D graphic attributes and visibility of the listed window parts by Object, Line Style, or Class. • Object: disables line style and weight controls for individual parts, and uses the line style and weight settings for the window object. • Line Style: controls line style and weight for the individual parts; these settings override the window object's settings. • Class: sets the line style and weight of individual parts by class. Select a class for each part to control its visibility and appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the window class to place the part in the same class as the window object.
Visibility Classes (Vectorworks Architect or Landmark required)	Select a class for each listed part to control its visibility and appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the window class to place the part in the same class as the window object.

Window Settings: 3D Visualization Pane

3D Visualization options are not enabled for the Opening **Sash** configuration.

Click to show/hide the parameters.

Parameter	Description
3D Hinge Direction Marker	
Show Interior/ Exterior Marker	Select to draw lines representing the hinge direction of the window sashes on the interior/ exterior side of the window in 3D
Use Hinge Marker Class Attributes	Select to set the hinge direction marker's attributes by class
Hinge Direction Marker Class	Select a class to control visibility and, if **Use Hinge Marker Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default window class to place the part in the same class as the window object.
Marker Points Towards	Set the hinge direction marker to point toward the hinge or toward the handle
Show 3D Open	Select to draw 3D window sashes as open at the specified **Open Angle**
Clerestory	Enabling **Clerestory** draws the window as dashed if the sill is higher than the specified elevation and draws the window as non-breaking (solid) if the sill is below the specified **Dash Height**

Window Settings: ID Tag Pane

Click to show/hide the parameters.

Parameter	Description
Include on Schedule	Select whether to include this window's information in the window schedule and also set the window ID tag to visible
ID Prefix	Assigns alphanumeric information before the numerical label value; adding prefix information is optional
ID Label	Assigns a numerical value to the ID; this number increments automatically if the auto-increment option is chosen in the ID Settings dialog box (see "Using the ID Label Tool" on page 1262)
ID Suffix	Assigns alphanumeric information after the label value; adding suffix information is optional
ID Class	Select a class to control visibility and, if **Use ID Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default window class to place the part in the same class as the window object.
Use ID Class Attributes	Select to use the ID class attributes for the bubble line, leader line, and leader line marker
Keep ID Horizontal	When selected, automatically rotates the ID so that it is horizontal
Bubble Shape	Select the ID tag bubble shape
Bubble Size	Enter the minimum bubble size (this value represents the bubble size times the layer scale; the bubble shape is maintained relative to the text inside it for ID bubble uniformity throughout the drawing file)
Bubble Line Attributes	Select the bubble line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness

Parameter	Description
Show Leader	Select to draw a leader line from the ID tag to the window object
Line Attributes	Select the leader line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness
Include Marker	Select whether to include a marker for the ID leader line and choose the desired style from the marker list, or select **Custom** to create a custom marker. Select **Edit Marker List** to open the Edit Marker List dialog box; see “Editing the Marker List” on page 1103.
Show Tag in 3D View (Vectorworks Architect or Landmark required)	Select to display the ID tag in 3D views
Horizontal Offset	If **Show Tag in 3D View** is selected, enter the horizontal offset of the 3D ID tag relative to the bottom left corner of the rough opening when looking at the exterior side of the window (a positive value moves the tag inside the opening relative to the starting corner; a negative value moves the tag outside the opening relative to the starting corner) The ID tag is offset slightly from the outside face of the window sash. The size and shape of the text and bubble, if present, will match the plan settings, and the text will always be right reading from the outside of the wall. The bubble will scale with text scaling in viewports.
Vertical Offset	If **Show Tag in 3D View** is selected, enter the vertical offset of the 3D ID tag relative to the bottom left corner of the rough opening when looking at the exterior side of the window (a positive value moves the tag inside the opening relative to the starting corner; a negative value moves the tag outside the opening relative to the starting corner)

Window Settings: Corner Window Pane

Vectorworks Architect or Landmark product required. Corner Window options are not enabled for non-rectangular window shapes or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Corner Window	Select this option to redraw the current window as a corner window. Settings that do not apply to a corner window may be disabled or changed when the corner window option is selected. When you click **OK** to exit the Window Settings dialog box, the window is moved to the corner of the wall to which it is most closely located. To complete the corner window, insert another one into the adjacent, attached wall. Corner windows do not work in tandem with one another. When you make adjustments to one window, you will also need to update the other. This can be accomplished quickly with the **Eyedropper** tool or from the Object Info palette. When you make adjustments to the wall containing a corner window, the window moves along with the wall and remains locked in the corner position. When adjusting the angle of the wall, the miter is automatically adjusted in most cases; if a corner window object does not automatically update, temporarily change a parameter in the Object Info palette to force a reset of the window object.

Parameter	Description
Corner Condition	Select Flush Glass to draw the glass right to the edge of the wall and wrap it around the corner, select Mitered Sash to draw the window with a mitered sash in the corner, select Corner Post to draw the window with a post in the corner, or select Opening to create a simple opening in a wall for defining a window break without creating any geometry to represent the window (the **Sash** configuration is automatically also changed to Opening). For windows with a corner post, specify the **Size** of the post and its **Offset** within the wall.

Window Settings: Jamb and Sash Pane

Jamb and Sash options are not enabled for the Opening **Sash** configuration.

Click to show/hide the parameters.

Parameter	Description
Jamb	
Jamb Width	Enter the face width of the window jamb (parallel to the wall)
Jamb Depth	Enter the depth of the window jamb (perpendicular to the wall)
Use Wall Depth	Select to set the jamb depth to the overall wall depth
Use Jamb Extension (Vectorworks Architect or Landmark required)	Select this field to specify a jamb extension to fill any gap between the interior face of the jamb and the interior face of the wall; enter the desired **Jamb Extension Width** In 2D views, extensions use the **Jamb** line styles and weights; in 3D views, the extension fill color and/or textures and visibility are controlled by the **Interior Jamb** class.
Sash	
Sash Width	Enter the sash width
Sash Depth	Enter the sash depth
Sash Offset (Vectorworks Architect or Landmark required)	Enter the offset of the sash relative to the jamb depth (a positive value moves the sash toward the window exterior; a negative value moves the sash toward the window interior)
Sash Width Offset (Vectorworks Architect or Landmark required)	Enter the sash width offset relative to the jamb width (leaving the default value of 0 keeps the sash adjacent to the jamb, and a positive value makes the sash intersect the jamb; a negative value is not permitted)
Glass Offset (Vectorworks Architect or Landmark required)	Enter the glass offset relative to the sash depth (a positive value moves the glass toward the window exterior; a negative value moves the glass toward the window interior)

Parameter	Description
Unequal Sash	To create a double hung window with an unequal sash height, select to customize the top sash height, and enter the desired scale factor for the sash. Select to customize the width of the left and right sliders for a picture window slider, and enter the desired scale factor for the sashes. The sash scale factor applies to both the left and right sliders.
Shim Gap	Enter the distance between the jamb exterior face and the rough opening (for the Opening **Sash** configuration, the **Shim Gap** is automatically set to 0)
Show Shim Gap in Plan (Vectorworks Architect or Landmark required)	Select whether to display the shim gap in Top/Plan view
Masonry Module	Enter the masonry module dimension

Window Settings: Sill Pane

Sill options are not enabled for the Opening **Sash** configuration or for non-rectangular window shapes, or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Include Sill	Select whether to include a sill and select the sill style type
Timber	Specify the sill lip thickness, amount of keep for the sill, sill height, total sill depth, sill extension width, stool lip thickness, and stool nose dimension
Masonry (Vectorworks Architect or Landmark required)	Specify the sill lip thickness, amount of keep for the sill, rise dimension, sill height, total sill depth, and sill extension width
Brick (Vectorworks Architect or Landmark required)	Specify the sill lip thickness, rise dimension, total sill depth, sill extension width, stool lip thickness, and stool nose dimension

Window Settings: Transom Pane

Transom options are not enabled for the Opening **Sash** configuration or for non-rectangular window shapes, or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Sash	
Sash Width	Enter the transom sash width
Sash Depth	Enter the transom sash depth

Parameter	Description
Mullion	
Mullion Width	Enter the transom mullion width
Mullion Depth	Enter the transom mullion depth
Muntins	
Pattern	Select the transom muntin pattern
Vertical/Horizontal Bars	For colonial muntins, enter the number of vertical/horizontal muntin bars Prairie and renaissance muntins come with pre-set configurations; therefore, any number of **Vertical/Horizontal Bars** specified are disregarded
Bar Depth/Width	Enter the transom muntin bar width/depth
Bar Offset	Enter the transom muntin bar offset
Number of Rays	For sunburst patterns, enter the number of muntin rays

Window Settings: Trim Pane

Trim options are not enabled for the Opening **Sash** configuration or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Include Interior/ Exterior Trim	Select whether to include interior/exterior trim
Width/Depth	Enter the interior/exterior trim width and depth
Include Trim Under Stool/Sill	Select whether to include interior/exterior trim along the window bottom

Window Settings: Lintel Pane

Vectorworks Architect or Landmark product required. Lintel options are not enabled for the Opening **Sash** configuration or for non-rectangular window shapes, or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Include Lintel	Select to add a lintel above the window or transom
Int./Ext. Protrusion	Enter the interior and exterior lintel protrusion
(1) Thickness	Enter the lintel thickness
(2) Angle	Enter the lintel angle
(3) Drop	Enter the length of the lintel drop

Window Settings: Muntins Pane

Muntin options are not enabled for the Opening **Sash** configuration or for non-rectangular window shapes.

Click to show/hide the parameters.

Parameter	Description
Top Sash/Bottom Sash	Select which sashes should have muntins applied to them
Pattern	Select the muntin pattern
Vertical/Horizontal Bars	For colonial muntins, enter the number of vertical/horizontal muntin bars Prairie and renaissance muntins come with pre-set configurations; therefore, any number of **Vertical/Horizontal Bars** specified are disregarded
Bar Depth/Width	Enter the muntin bar width/depth
Bar Offset	Enter the muntin bar offset
Number of Rays	For sunburst patterns, enter the number of muntin rays
Custom Top Sash Muntins (Vectorworks Architect or Landmark required)	For a double hung window with an unequal sash and colonial muntins, select to customize the top sash, and enter the desired number of horizontal bars

Window Settings: Interior Shutters Pane

Vectorworks Architect or Landmark product required. Interior Shutter options are not enabled for the Opening **Sash** configuration or for non-rectangular window shapes, or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Include Interior Shutters	Select whether to include interior shutters
Use Custom Width	If Square is selected as the **Top Shape**, select whether the interior shutters should be a width other than the default value (1/2 the width of the window) and enter the desired custom width
Frame Depth	Enter the depth for the interior shutters
Offset from Jamb	Enter the offset distance from the interior edge of the window jamb that shutters will be positioned when fully opened
Panel Shutter	Select to create paneled shutters
Panel Frame Width	Enter the rail and stile width of the paneled shutter
Number of Panels	Select **One panel per sash** to divide shutters at sash divisions or select to divide panels equally by the **Custom number of panels** specified
Custom Shutter	Click **Choose Symbol** to select a custom shutter symbol from default content; see “Resource Libraries” on page 219 Custom shutters must be 3D-only symbols made from generic solids, saved in the default content location.

Window Settings: Exterior Shutters Pane

Vectorworks Architect or Landmark product required. Exterior Shutter options are not enabled for the Opening **Sash** configuration or for non-rectangular window shapes, or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Include Exterior Shutters	Select whether to include exterior shutters
Use Custom Width	If Square is selected as the **Top Shape**, select whether the exterior shutters should be a width other than the default value (1/2 the width of the window) and enter the desired custom width
Frame Depth	Enter the depth for the exterior shutters
Offset from Jamb	Enter the offset distance from the interior edge of the window jamb that shutters will be positioned when fully opened
Panel Shutter	Select to create paneled shutters
Panel Frame Width	Enter the rail and stile width of the paneled shutter
Number of Panels	Select **One panel per sash** to divide shutters at sash divisions or select to divide panels equally by the **Custom number of panels** specified
Custom Shutter	Click **Choose Symbol** to select a custom shutter symbol from default content; see “Resource Libraries” on page 219 Custom shutters must be 3D-only symbols made from generic solids, saved in the default content location.

Window Settings: Centerline Marker Pane

Vectorworks Architect or Landmark product required. Centerline Marker options are not enabled for curtain wall windows.

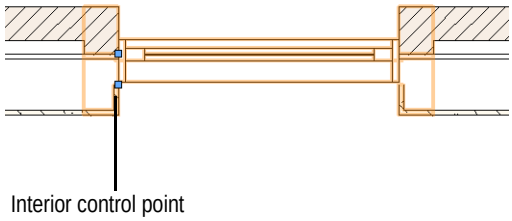
Click to show/hide the parameters.

Parameter	Description
Centerline Marker	Select to display a centerline marker in Top/Plan view.
Size	Set the size of the centerline
Class	Select a class to control visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the default window class to place the part in the same class as the window object.
Line Attributes	Select the line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness. If **Use Class Attributes** is selected, the line attributes are set by the part’s class
Include Marker	Select whether to include a marker and choose the desired style from the marker list, or select **Custom** to create a custom marker. Select **Edit Marker List** to open the Edit Marker List dialog box; see “Editing the Marker List” on page 1103. If **Use Class Attributes** is selected, the marker attributes are set by the part’s class.
Use Class Attributes	Select to set the centerline marker’s attributes (including marker style) by class rather than the parameters in the Window Settings dialog box

Window Settings: Interior Wall Detail Pane

Interior Wall Detail options are not enabled for non-rectangular window shapes or for curtain wall windows.

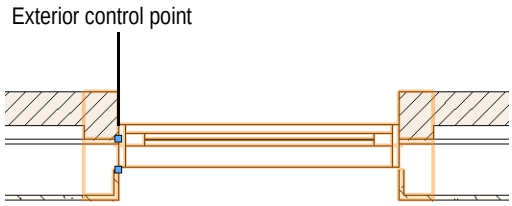
Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of interior wall components to wrap; when interior wall components are specified, the window enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the interior wall; the splay can be specified either as an angle/ diagonal or with width and depth values
(1) Splay Angle	Enter the interior wall splay angle
(2) Splay Diagonal	Enter the interior wall splay diagonal value
(3) Splay Width	Enter the interior wall splay width
(4) Splay Depth	Enter the interior wall splay depth

Window Settings: Exterior Wall Detail Pane

Exterior Wall Detail options are not enabled for non-rectangular window shapes or for curtain wall windows.

Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of exterior wall components to wrap; when exterior wall components are specified, the window enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the exterior wall; the splay can be specified either as an angle/ diagonal or with width and depth values
(1) Splay Angle	Enter the exterior wall splay angle
(2) Splay Diagonal	Enter the exterior wall splay diagonal value
(3) Splay Width	Enter the exterior wall splay width
(4) Splay Depth	Enter the exterior wall splay depth

Window Settings: Classes Pane

The visibility of the overall 3D window is controlled by the **Class** setting on the Object Info palette; part settings are controlled from the Window Settings dialog box.

Click to show/hide the parameters.

Parameter	Description
Window parts	Select a class for each window part to control its fill color and/or texture and visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the window class to place the part in the same class as the window object. The Vectorworks Architect and Landmark products have a more comprehensive list of window parts available to be assigned a class. For windows with an Opening **Sash** configuration, in a 3D view, a no fill/no pen weight extrude completely fills the wall in place of the window; this extrude is placed in the Interior Jamb class and is filtered out in IFC export operations. Change the class settings to control the visibility of the extrude, as needed.

Window Settings: Data Pane

Certain data fields represent calculated values and cannot be edited; as a result, the **Field Name** and **Field Value** appear dimmed for those data fields.

Click to show/hide the parameters.

Parameter	Description
Field Name	Select the data field from the list and its name displays beneath the list; information associated with the **Field Name** can be specified in the **Field Value** area. The user fields can include additional user-defined information with the window.
Field Value	Enter data for use in the window schedule

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Windows
Window Properties
Creating Window Schedules
Creating a Custom Window Sash Opening
Window Terminology Specific to the United Kingdom

A L Creating a Custom Window Sash Opening

In the Vectorworks Architect and Landmark products, a custom window sash opening can be created for selection in the Window Settings dialog box. A rectangular sash opening can contain any number of rectangular sashes.

To create a custom window sash opening:

1. From the General pane of the Window Settings dialog box, select the Custom **Sash** operation and click **Custom Sash Options**.

   The Custom Sash Options dialog box opens for selecting and specifying the sash opening parameters. As parameters are defined, the preview dynamically displays the configuration operations of the sashes filling the opening.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Num. of Rows/Columns	Specify the number of rows and columns in the sash opening (the total number of sashes cannot exceed 64)
Overall Width/Height	Displays the overall width and height of the custom sashes
Mullion Width/Depth	Displays the mullion width and depth
Selected Sash	The currently active sash displays in red to indicate that it is selected
Width/Height	For the active sash, specify the desired **Width** and **Height**
Operation	For the active sash, choose the desired window type
Include Muntins	Select whether to include muntins for each sash
Lock Sash Configuration	When selected, locks the sash dimensions and operation
Unequal Sash	To create a double hung window with an unequal sash height, select to customize the top sash height, and enter the desired scale factor for the sash; to customize the horizontal top sash muntins, see “Window Settings: Muntins Pane” on page 601. Select to customize the width of the left and right sliders for a picture window slider, and enter the desired scale factor for the sashes. The sash scale factor applies to both the left and right sliders.
<< or >>	Click **<<** or **>>** to move backward or forward to activate sashes for dimension and operation specification

2. Click **OK** to close the Custom Sash Options dialog box and return to the Window Settings dialog box. The Window Settings dialog box preview is updated with the customization results.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Window Settings: General Pane
Window Terminology Specific to the United Kingdom

Window Properties

To edit window parameters, click **Settings** from the Object Info palette, double-click the window to open the Window Settings dialog box, or right-click (Windows) or Ctrl-click (Mac) on a window and select **Edit** from the context menu.

When multiple window objects are selected for editing, and if the parameter settings of the selected objects are different, parameters display in an “indeterminate state.” Any values changed through the dialog box are changed for all the selected objects.

Window objects can also be edited in the Object Info palette. If the window has been inserted as a plug-in object, most settings from the Window Settings dialog box display. If the window is a black symbol made from a window object, fields pertaining to window geometry do not display. See “Symbol Types” on page 237.

For the Vectorworks Architect and Landmark products, most, but not all, parameters can be edited from both the Window Settings dialog box and the Object Info palette for ease of access. The fields in the Object Info palette are named similarly (but not always identically) to those in the Window Settings dialog box, and roughly reflect the order in which settings are entered in the dialog box, for ease of editing. Curtain wall windows have limited parameter options.

For the Vectorworks Fundamentals and Spotlight products, certain frequently used parameters are available to be edited directly in the Object Info palette, and do not necessarily reflect the order in which settings are entered in the dialog box. All other parameters are edited only from the Window Settings dialog box.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The window parameters are described in “Inserting Windows in Vectorworks Fundamentals” on page 587 and “Inserting Windows in Vectorworks Design Series” on page 594. Only the parameters that are different are described here.

Click to show/hide the parameters.

Parameter	Description
Flip	Click to flip the orientation of windows inserted in a wall
Set Position	Click to activate the **Move by Points** tool in Reference Point mode and move the window by a specified distance from a reference point

Inserting Windows in Vectorworks Fundamentals
Inserting Windows in Vectorworks Design Series
Creating New Symbols
Moving Objects
Window Terminology Specific to the United Kingdom

Creating Window Schedules

The Window Schedule can be added to the drawing from the **VA Create Schedule** command (see “Records and Schedules” on page 1859) or the Resource Browser. From the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder that is included with the Vectorworks Architect product (see “Resource Libraries” on page 219). Drag the Window Schedule worksheet to the drawing. The worksheet is populated with information from the window objects in the current drawing (note that corner windows are counted as two separate windows in the worksheet). To edit the worksheet after it has been created, see “Using Worksheets” on page 1319.

Users of the Vectorworks Landmark and Spotlight products can also create the worksheet by creating a report for objects with a window record. To create the report from Landmark select **Tools > Reports > Create Report**; from Spotlight select **Spotlight > Reports > Create Report.**

Inserting Windows in Vectorworks Fundamentals
Inserting Windows in Vectorworks Design Series
Moving Objects
Window Terminology Specific to the United Kingdom

Window Terminology Specific to the United Kingdom

Certain window terminology varies in the United Kingdom. The window object is automatically localized in the Vectorworks program for users with a United Kingdom serial number; however, since the documentation is not localized, the terminology does not match. This table shows how the terms are mapped.

U.S. Term (in Documentation)	U.K. Term (in Vectorworks Program)
Transom	Fanlight
Rough Opening	Structural Opening
Muntins (Leaf and Transom)	Glazing Bars
Colonial (Muntin Style)	Georgian
Prairie w/sunburst (Muntin Style)	Prairie w/ Fan Top
Colonial w/sunburst (Muntin Style)	Georgian w/Fan Top

U.S. Term (in Documentation)	U.K. Term (in Vectorworks Program)
Mullion (Transom)	Transom Bar
Trim	Architraves
Interior Trim	Interior Architraves
Exterior Trim	Exterior Architraves
Masonry Opening (Data)	Nearest Full-Module of Masonry
Window Configurations	Window Configurations
Fixed Glass	Fixed Light
Single Hung	Sliding Sash (lower)
Double Hung	Sliding Sash (upper and lower)
Casement	Side-Hung Casement
Bi-Parting Casement	Double-Hung Casement
Awning	Top-Hung Casement
Hopper	Bottom-Hung Casement

Inserting Windows in Vectorworks Fundamentals
Inserting Windows in Vectorworks Design Series
Creating a Custom Window Sash Opening

Doors

In the Vectorworks Design Series products, the door object is inserted with the **Door** tool. In the Vectorworks Fundamentals product, the door object is inserted from an object library through the Resource Browser. All Vectorworks products include a door object containing parameters similar to those described here. The Vectorworks Architect and Landmark products include enhanced capabilities.

The Vectorworks Architect and Landmark products also include door symbols in standard sizes and various configurations, ready to install in walls. These products also include the ability to insert doors into curtain walls as special "curtain wall doors." Additionally, the Vectorworks Architect product provides extensive libraries of door symbols, listed by configuration and size.

> The **Update Plug-in Objects** command may need to be run on files containing doors that were created in an earlier version of the Vectorworks Architect or Landmark product. This command converts the doors to the latest format; see "Migrating from Previous Versions" on page 28.

A door can be customized by adjusting various parameters, and then saved as a symbol so that its parameters are preset upon insertion. This eliminates the need to repeatedly apply parameters, maximizes memory efficiency, and allows global editing of symbols. See "Creating New Symbols" on page 239.

If a custom door is unique (that is, it has only one instance) in a project, creating a symbol from it is unnecessary. However, making a symbol of the installed door makes it easy to reuse it in other drawings should there be a need to do so in the future.

After the symbol is created, it can be selected from the Door Settings dialog box (by selecting **Use Symbol Geometry** as described in "Door Settings: General Pane" on page 618), where it can take advantage of functionality such as embedded IDs and data, cavity wrapping, splays, and wall offsets. To enable automatic ID labeling, from the ID Tag pane of the settings dialog box, select **Include on Schedule**, and specify the ID parameters. If necessary, set the class to which the ID is assigned to visible. Alternatively, set the automatic ID labeling through the **ID Label** tool; see "Using the ID Label Tool" on page 1262.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Doors in Vectorworks Fundamentals

To insert a door in Vectorworks Fundamentals:

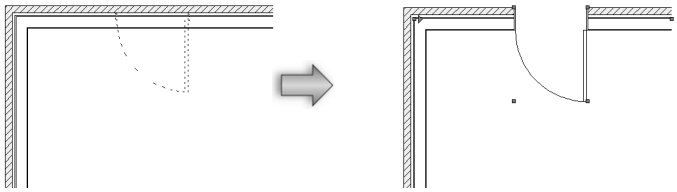
1. Insert a door symbol from the Resource Browser (see "Inserting Symbols" on page 242). The direction of a door inserted in a wall can be changed later by clicking **Flip** on the Object Info palette or from the context menu.

   A library with an editable door symbol is included with the Vectorworks Fundamentals product. If this content was not downloaded during installation, download it at any time by selecting **Help > Download Content** (see "Resource Libraries" on page 219).

   The **Door** tool is available for Vectorworks Fundamentals software but is not present in the Fundamentals workspace. It can be added to the Fundamentals workspace (see "Creating or Editing Workspaces" on page 1835) and subsequently used to insert door objects.

2. Click the **Settings** button on the Object Info palette to open the Door Settings dialog box. Select each pane and specify the door parameters. Click **OK** to set the door parameters and close the dialog box.

   Several features of the door are described as "interior" or "exterior." These include trim and wall-wrap parts. These elements are determined based on the internal and external faces of the wall. The left side of the wall (as viewed along the wall direction) is always "exterior," and the right side is "interior" (see "Wall Direction" on page 506). Flipping the door does not flip these elements.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Door Settings: Preview

Click to show/hide the parameters.

Parameter	Description
3D Preview	Dynamically displays a 3D preview of the door with the currently selected parameters
View	Select the standard view for the 3D preview of the door
Render	Select the render mode for the 3D preview of the door
Top/Plan Preview	Dynamically displays a Top/Plan preview of the door with the currently selected parameters

Door Settings: General Pane

Click to show/hide the parameters.

Parameter	Description
Size Reference: Leaf Size	For a leaf size door, the width/height of the door leaf is measured at the inside of the jamb; set the **Width** and **Height**
Use Symbol Geometry	Select **Use Symbol Geometry** and click **Browse** to select a door symbol from the current file's resources. Select a symbol from the graphical list of **Symbols** and click **OK**. The symbol **Name** displays in the Door Settings dialog box and the preview is updated with the selected symbol. When a symbol is selected, all fields pertaining to the door's geometry are disregarded; however, parametric values are still available for scheduling and reporting purposes only.
Top Shape	Select the shape of the top of the door
Transom	Select whether to add a transom above the door; transom parameters are set from the Transom pane
Rise	Enter the distance between the start of the top shape to the top of the door (not applicable for Square and Round **Top Shape**)
Spring	Enter the distance traveled above the floor before the **Top Shape** begins
Offset in Wall	Enter the distance to offset the door insertion point from the centerline of the wall
Configuration	Select the general door configuration. Depending on the selection, a variety of door settings are enabled or disabled automatically, as described in the following sections.
Operation	Select the advanced operation for a sliding or complex swing door, where "O" represents a fixed door and "X" represents an operating door

Door Settings: 2D Visualization Pane

Click to show/hide the parameters.

Parameter	Description
Show Plan Detail	Select to display all the door details in Top/Plan view
Draw Wall Lines	Select whether to draw wall lines at all times; wall lines are drawn in the Ceiling-Main class to easily create reflected ceiling plans
Set Attributes By	Select whether to set the 2D graphic attributes and visibility of the listed door parts by Object, Line Style, or Class. • Object: disables line style and weight controls for individual parts, and uses the line style and weight settings for the door object. • Line Style: controls line style and weight for the individual parts; these settings override the door object's settings. • Class: sets the line style and weight of individual parts by class. Select a class for each part to control its visibility and appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the door class to place the part in the same class as the door object.

Door Settings: 3D Visualization Pane

3D Visualization options are not enabled for the Opening **Configuration.**

Click to show/hide the parameters.

Parameter	Description
3D Hinge Direction Marker	Select to draw lines representing the 3D hinge direction
Show Interior/ Exterior Marker	Select to draw lines representing the hinge direction of the door leaves on the interior/ exterior side of the door in 3D
Use Hinge Marker Class Attributes	Select to set the hinge direction marker's attributes by class
Hinge Direction Marker Class	Select a class to control visibility and, if **Use Hinge Marker Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default door class to place the part in the same class as the door object.
Marker Points Towards	Set the hinge direction marker to point toward the hinge or toward the handle
Draw 3D Open	Select to draw 3D doors as open at the specified **Open Angle**
3D Detail Level	Select the level of detail to include for 3D doors. In Low and Medium settings, some door details are not drawn in 3D views; these settings simplify the appearance for viewing the drawing at a small scale.

Door Settings: ID Tag Pane

Click to show/hide the parameters.

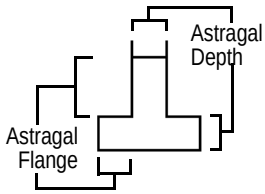
Parameter	Description
Include on Schedule	Select whether to include this door's information in the door schedule and also set the door ID tag to visible; see "Creating Door Schedules or Worksheets" on page 631
ID Prefix	Assigns alphanumeric information before the numerical label value; adding prefix information is optional
ID Label	Assigns a numerical value to the ID; this number increments automatically if the auto-increment option is chosen in the ID Settings dialog box (see "Using the ID Label Tool" on page 1262)
ID Suffix	Assigns alphanumeric information after the label value; adding suffix information is optional
ID Class	Select a class to control visibility and, if **Use ID Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default door class to place the part in the same class as the door object.
Use ID Class Attributes	Select to use the ID class attributes for the bubble line, leader line, and leader line marker
Keep ID Horizontal	When selected, automatically rotates the ID so that it is horizontal
Bubble Shape	Select the ID tag bubble shape
Bubble Size	Enter the minimum bubble size (this value represents the bubble size times the layer scale; the bubble shape is maintained relative to the text inside it for ID bubble uniformity throughout the drawing file)
Bubble Line Attributes	Select the bubble line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness
Show Leader	Select to draw a leader line from the ID tag to the door object
Line Attributes	Select the leader line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness
Include Marker	Select whether to include a marker for the ID leader line and choose the desired style from the marker list, or select **Custom** to create a custom marker. Select **Edit Marker List** to open the Edit Marker List dialog box; see "Editing the Marker List" on page 1103.

Door Settings: Jamb Pane

Jamb options are not enabled for the Opening **Configuration**.

Click to show/hide the parameters.

Parameter	Description
Width	Enter the face width of the door jamb (parallel to the wall)
Depth	Enter the depth of the door jamb (perpendicular to the wall)
Use Wall Depth	Select to set the jamb depth to the overall wall depth
Shim Gap	Enter the distance between the jamb exterior face and the rough opening (for the Opening **Configuration**, the **Shim Gap** is automatically set to 0)
Masonry Module	Enter the masonry module dimension

Parameter	Description
Astragal Flange / Depth	Use these two dimensions to determine the size of the astragal flange or depth (enter zero to display without an astragal flange); enabled if Swing Bi-part is selected in the **Configuration** field on the General pane 

Door Settings: Leaf Pane

Leaf options are not enabled for the Opening **Configuration.**

Click to show/hide the parameters.

Parameter	Description
Leaf	Select the desired leaf type
Solid	
Thickness	Enter the thickness of the door slab
Top Rail Width	Enter the width of the top door rail
L/R Stile Width	Enter the width of the left and right stile
Bottom Rail Width	Enter the width of the bottom door rail
Vertical/Horizontal Panels	Enter the number of vertical/horizontal panels in the door
Mid Stile Width	Enter the width of any interim stiles
Set Top Panel Height	Select to make the top door panel a different height than the other panels and specify the desired **Panel Height**
Glass	The Glass leaf type enables the options specific to glass doors on this pane, such as Muntin Style; it does not make the door leaves render as transparent glass. To make the glass render properly, apply a **Glazing** class on the Classes pane.
Thickness	Enter the thickness of the door slab
Top Rail Width	Enter the width of the top door rail
L/R Stile Width	Enter the width of the left and right stile
Bottom Rail Width	Enter the width of the bottom door rail
Muntin Style	Select the muntin style
Vertical/Horizontal Bars	Enter the number of vertical/horizontal muntin bars
Bar Width/Depth	Enter the muntin bar width/depth
Panel	
Thickness	Enter the thickness of the door slab
Top Rail Width	Enter the width of the top door rail

Parameter	Description
L/R Stile Width	Enter the width of the left and right stile
Bottom Rail Width	Enter the width of the bottom door rail
Vertical/Horizontal Panels	Enter the number of vertical/horizontal panels in the door
Mid Stile Width	Enter the width of any interim stiles
Set Top Panel Height	Select to make the top door panel a different height than the other panels and specify the desired **Panel Height**

Door Settings: Lights Pane

Lights options are not enabled for the Opening **Configuration**.

Click to show/hide the parameters.

Parameter	Description
Include Sidelights	Select to include sidelights
Left/Right Width	Enter the left/right sidelight width
Sash Width	Enter the sidelight and transom sash width
Sash Depth	Enter the sidelight sash depth
Mullion Width	Enter the sidelight mullion width
Mullion Depth	Enter the sidelight mullion depth
Include Muntins	Select whether to add muntins to the sidelights
Style	Select the sidelight muntin style
Vertical/Horizontal Bars	Enter the number of vertical/horizontal muntin bars
Bar Width	Enter the sidelight muntin bar width

Door Settings: Transom Pane

Transom options are not enabled for the Opening **Configuration**.

Click to show/hide the parameters.

Parameter	Description
Sash Width	Enter the transom sash width
Sash Depth	Enter the transom sash depth
Mullion Width	Enter the transom mullion width
Mullion Depth	Enter the transom mullion depth
Include Muntins	Select whether to add muntins to the transom
Style	Select the transom muntin style
Vertical/Horizontal Bars	Enter the number of vertical/horizontal muntin bars

Parameter	Description
Bar Width	Enter the transom muntin bar width
Bar Offset	Enter the muntin bar offset for Prairie muntins

Door Settings: Trim Pane

Trim options are not enabled for the Opening **Configuration**.

Click to show/hide the parameters.

Parameter	Description
Include Interior/ Exterior Trim	Select whether to include interior/exterior trim
Width/Depth	Enter the interior/exterior trim width and depth

Door Settings: Hardware Pane

Hardware options are not enabled for the Opening **Configuration.**

Click to show/hide the parameters.

Parameter	Description
Include Hardware	Select to include a door hardware set with the door object; see “Assigning, Creating, Editing, and Deleting Door Hardware Sets” on page 629.
View Hardware	Click to open the View Door Hardware Set dialog box to view the parameters for the currently selected door hardware set; click **OK** to return to the Door Settings dialog box
Manage Hardware	Click to open the Door Hardware Library dialog box to assign, create, edit, or delete door hardware sets

Door Settings: Kick Plates Pane

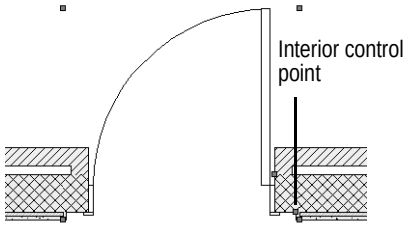
Kick Plates options are not enabled for the Opening **Configuration** or for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Interior/Exterior Kick Plate	Select to include a kick plate on either side of the leaf
Height	Set the height of the kick plate While the height is variable, the kick plate’s distance from the right, left and bottom edges of the door is a fixed value of one inch/25.4 mm.
Class	Select a class to control visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the default door class to place the part in the same class as the door object.

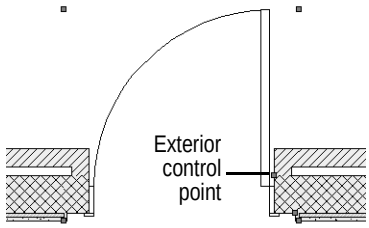
Door Settings: Interior Wall Detail Pane

Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of interior wall components to wrap; when interior wall components are specified, the door enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the interior wall; the splay can be specified either as an angle/ diagonal or with width and depth values
(1) Angle	Enter the interior wall splay angle
(2) Diag	Enter the interior wall splay diagonal value
(3) Width	Enter the interior wall splay width
(4) Depth	Enter the interior wall splay depth

Door Settings: Exterior Wall Detail Pane

Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of exterior wall components to wrap; when exterior wall components are specified, the door enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the exterior wall; the splay can be specified either as an angle/ diagonal or with width and depth values
(1) Angle	Enter the exterior wall splay angle
(2) Diagonal	Enter the exterior wall splay diagonal value
(3) Width	Enter the exterior wall splay width
(4) Depth	Enter the exterior wall splay depth

Door Settings: Classes Pane

The visibility of the overall 3D door is controlled by the **Class** setting on the Object Info palette; part settings are controlled from the Door Settings dialog box.

Click to show/hide the parameters.

Parameter	Description
Door parts	Select a class for each door part to control its visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the door class to place the part in the same class as the door object. The Architect and Landmark products have a more comprehensive list of door parts available to be assigned a class. For doors with an Opening **Configuration**, in a 3D view a no fill/no pen weight extrude completely fills the wall in place of the door; this extrude is placed in the Interior Jamb class. Change the class settings to control the visibility of the extrude, as needed

Door Settings: Data Pane

Certain data fields represent calculated values and cannot be edited; as a result, the **Field Name** and **Field Value** appear dimmed for those data fields.

Click to show/hide the parameters.

Parameter	Description
Field Name	Select the data field from the list and its name displays beneath the list; information associated with the **Field Name** can be specified in the Field Value area. The user fields can include additional user-defined information with the door.
Field Value	Enter data for use in the door schedule

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Doors
Inserting Doors in Vectorworks Design Series
Door Properties
Door Terminology Specific to the United Kingdom
Editing Symbols in Walls

Inserting Doors in Vectorworks Design Series

To insert a door in Vectorworks Design Series:

1. Select the **Door** tool from the Building Shell tool set.

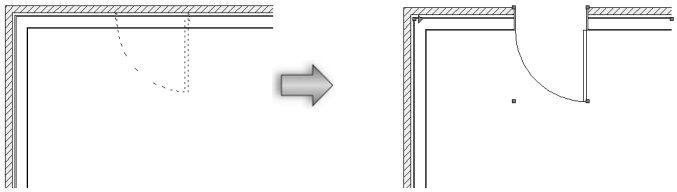
   Alternatively, if placing a curtain wall door into a curtain wall (Vectorworks Architect or Landmark required), select a panel with the **Edit Curtain Wall** tool, and then right-click (Windows) or Ctrl-click (Mac) on the panel and select **Insert Door** from the context menu. The door is automatically inserted as a curtain wall door.

2. If this is the first time a door is inserted into the drawing, or if you wish to change the default preferences that apply to subsequently placed doors, click **Preferences** from the Tool bar.

   The Door Settings dialog box opens. The settings are grouped into several panes of related parameters, which are listed on the left side of the dialog box. Select each pane and specify the door parameters. Click **OK** to set the door parameters and close the dialog box. These parameters can be edited for placed doors from the Object Info palette, as described in “Door Properties” on page 631.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Click in the drawing area or in a wall to set the insertion point of the door, and click again to set the rotation. The direction of a door inserted in a wall can be changed later by clicking **Flip** on the Object Info palette or from the context menu.

   Several features of the door are described as "interior" or "exterior." These include trim and wall-wrap parts. These elements are determined based on the internal and external faces of the wall. The left side of the wall (as viewed along the wall direction) is always "exterior," and the right side is "interior" (see "Wall Direction" on page 506). Flipping the door does not flip these elements.



Door Settings: Preview

Click to show/hide the parameters.

Parameter	Description
3D Preview	Dynamically displays a 3D preview of the door with the currently selected parameters
View	Select the standard view for the 3D preview of the door
Render	Select the render mode for the 3D preview of the door
Top/Plan Preview	Dynamically displays a Top/Plan preview of the door with the currently selected parameters

Door Settings: General Pane

Click to show/hide the parameters.

Parameter	Description
Size Reference	Select the appropriate option to define the door width and height; the **Size Reference** selection automatically updates the **Width** and **Height** fields, but does not change the size of the door: • Leaf Size - this represents the door width and height that measures to the inside of the jamb, excluding the jamb, transom, and sidelights; this is the default setting for imperial units • Unit Size - this represents the current overall door width and height that measures to the outside of the jamb, including the jamb, transom, and sidelights • Rough Opening - this represents the current overall door width and height that measures to the outside of the jamb, including the jamb, transom, sidelights, and shim gap (for the Opening **Configuration**, the Size Reference is automatically changed to Rough Opening); this is the default setting for metric units Vectorworks Architect or Landmark product required for Unit Size and Rough Opening.
Width/Height	The width/height of the door is measured based on the **Size Reference** selection: • Leaf size - width/height of the door leaf is measured at the inside of the jamb • Unit Size - width/height is measured at the outside of the jamb • Rough Opening - width/height measures the rough opening

Parameter	Description
Use Symbol Geometry	Select **Use Symbol Geometry** and click **Browse** to select a door symbol from the current file's resources. Select a symbol from the graphical list of **Symbols** and click **OK**. The symbol **Name** displays in the Door Settings dialog box and the preview is updated with the selected symbol. When a symbol is selected, all fields pertaining to the door's geometry are disregarded; however, parametric values are still available for scheduling and reporting purposes only.
Top Shape	Select the shape of the top of the door
Transom	Select whether to add a transom above the door; transom parameters are set from the Transom pane
Rise	Enter the distance between the start of the top shape to the top of the door (not applicable for Square and Round **Top Shape**)
Spring	Enter the distance traveled above the floor before the **Top Shape** begins
Offset in Wall	Enter the distance to offset the door insertion point from the centerline of the wall
Configuration	Select the general door configuration. To create a simple opening in a wall for defining a door break without creating any geometry to represent the door, select the Opening option (Vectorworks Architect or Landmark required). Depending on the selection, a variety of door settings are enabled or disabled automatically, as described in the following sections.
Operation	Select the advanced operation for a sliding or complex swing door, where "O" represents a fixed door and "X" represents an operating door
Unequal Leaf (Vectorworks Architect or Landmark required)	For bi-part and double acting configurations, select this option to insert a door with leaves of unequal widths
Width Leaf 1/2	When **Unequal Leaf** is selected, enter the width of the first door leaf; the width of the second door leaf is calculated automatically and displayed in as the **Width Leaf 2** value
Curtain Wall Door (Vectorworks Architect or Landmark required)	Indicates that the door will be placed in a curtain wall, where its width and height are based on the surrounding frames where it is placed. A curtain wall door does not have as many available parameter options as a standard door. Configurations are limited to those that apply to curtain wall placement.

Door Settings: 2D Visualization Pane

Click to show/hide the parameters.

Parameter	Description
Show Plan Detail	Select to display all the door details in Top/Plan view
Draw Wall Lines	Select whether to draw wall lines at all times; wall lines are drawn in the Ceiling-Main class to easily create reflected ceiling plans

Parameter	Description
Set Attributes By	Select whether to set the 2D graphic attributes and visibility of the listed door parts by Object, Line Style, or Class. • Object: disables line style and weight controls for individual parts, and uses the line style and weight settings for the door object. • Line Style: controls line style and weight for the individual parts; these settings override the door object's settings. • Class: sets the line style and weight of individual parts by class. Select a class for each part to control its visibility and appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the door class to place the part in the same class as the door object.
Visibility Classes (Vectorworks Architect or Landmark required)	Select a class for each listed part to control its visibility and appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the door class to place the part in the same class as the door object.

Door Settings: 3D Visualization Pane

3D Visualization options are not enabled for the Opening **Configuration.**

Click to show/hide the parameters.

Parameter	Description
3D Hinge Direction Marker	Select to draw lines representing the 3D hinge direction
Show Interior/ Exterior Marker	Select to draw lines representing the hinge direction of the door leaves on the interior/ exterior side of the door in 3D
Use Hinge Marker Class Attributes	Select to set the hinge direction marker's attributes by class
Hinge Direction Marker Class	Select a class to control visibility and, if **Use Hinge Marker Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default door class to place the part in the same class as the door object.
Marker Points Towards	Set the hinge direction marker to point toward the hinge or toward the handle
Draw 3D Open	Select to draw 3D doors as open at the specified **Open Angle**
3D Detail Level	Select the level of detail to include for 3D doors. In Low and Medium settings, some door details are not drawn in 3D views; these settings simplify the appearance for viewing the drawing at a small scale.

Door Settings: ID Tag Pane

Click to show/hide the parameters.

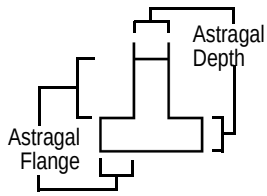
Parameter	Description
Include on Schedule	Select whether to include this door's information in the door schedule and also set the door ID tag to visible; see "Creating Door Schedules or Worksheets" on page 631

Parameter	Description
ID Prefix	Assigns alphanumeric information before the numerical label value; adding prefix information is optional
ID Label	Assigns a numerical value to the ID; this number increments automatically if the auto-increment option is chosen in the ID Settings dialog box (see “Using the ID Label Tool” on page 1262)
ID Suffix	Assigns alphanumeric information after the label value; adding suffix information is optional
ID Class	Select a class to control visibility and, if **Use ID Class Attributes** is selected, appearance. The classes present in the drawing are listed; alternatively, create a new class. Select the default door class to place the part in the same class as the door object.
Use ID Class Attributes	Select to use the ID class attributes for the bubble line, leader line, and leader line marker
Keep ID Horizontal	When selected, automatically rotates the ID so that it is horizontal
Bubble Shape	Select the ID tag bubble shape
Bubble Size	Enter the minimum bubble size (this value represents the bubble size times the layer scale; the bubble shape is maintained relative to the text inside it for ID bubble uniformity throughout the drawing file)
Bubble Line Attributes	Select the bubble line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness
Show Leader	Select to draw a leader line from the ID tag to the door object
Line Attributes	Select the leader line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness
Include Marker	Select whether to include a marker for the ID leader line and choose the desired style from the marker list, or select **Custom** to create a custom marker. Select **Edit Marker List** to open the Edit Marker List dialog box; see “Editing the Marker List” on page 1103.
Show Tag in 3D View (Vectorworks Architect or Landmark required)	Select to display the ID tag in 3D views
Horizontal Offset	Enter the horizontal offset of the 3D ID tag relative to the bottom left corner of the rough opening when looking at the exterior side of the door (a positive value moves the tag inside the opening relative to the starting corner; a negative value moves the tag outside the opening relative to the starting corner) The ID tag is offset slightly from the outside face of the door leaf. The size and shape of the text and bubble, if present, will match the plan settings, and the text will always be right reading from the outside of the wall. The bubble will scale with text scaling in viewports.
Vertical Offset	Enter the vertical offset of the 3D ID tag relative to the bottom left corner of the rough opening when looking at the exterior side of the door (a positive value moves the tag inside the opening relative to the starting corner; a negative value moves the tag outside the opening relative to the starting corner)

Door Settings: Jamb Pane

Jamb options are not enabled for the Opening **Configuration** or for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Width	Enter the face width of the door jamb (parallel to the wall)
Depth	Enter the depth of the door jamb (perpendicular to the wall)
Use Wall Depth	Select to set the jamb depth to the overall wall depth
Use Jamb Extension (Vectorworks Architect or Landmark required)	Select this field to specify a jamb extension to fill any gap between the interior face of the jamb and the interior face of the wall; enter the desired **Jamb Extension Width** In 2D views, extensions use the **Jamb** line styles and weights; in 3D views, the extension fill color and/or textures and visibility are controlled by the **Int. Jamb** class.
Shim Gap	Enter the distance between the jamb exterior face and the rough opening (for the Opening **Configuration**, the **Shim Gap** is automatically set to 0)
Show Shim Gap in Plan (Vectorworks Architect or Landmark required)	Select whether to display the shim gap in Top/Plan view
Masonry Module	Enter the masonry module dimension
Astragal Flange / Depth	Use these two dimensions to determine the size of the astragal flange or depth (enter zero to display without an astragal flange); enabled if Swing Bi-part is selected in the **Configuration** field on the General pane 

Door Settings: Leaf Pane

Leaf options are not enabled for the Opening **Configuration.**

Click to show/hide the parameters.

Parameter	Description
Leaf	Select the desired leaf type
Solid	
Thickness	Enter the thickness of the door slab
Include Vision Panels (Vectorworks Architect or Landmark required)	Select to include vision panel(s)

Parameter	Description
Vision Panel Shape	Select the shape of the vision panel(s)
Bottom Offset	Enter an offset value, measured from the bottom of the leaf to the bottom of the lowest vision panel
Right Offset	Enter an offset value, measured from the edge of the leaf to the nearest edge of the vision panel. For the most common door configurations, this parameter is measured from the unhinged edge of the leaf, which is the right edge in Vectorworks' default front view in the 3D preview. However, with some of the more complex, multi-part door configurations, the offset is measured from different points to provide correct alignment of vision panels. Consult the 3D Preview to confirm the appearance.
Panel Height/ Width	Enter the height/width of the individual vision panel(s) **Panel Width** is enabled only for the rectangle and oval vision panel shapes. For square and round, the width is automatically set equal to the height.
Number of Panels	Enter the number of vision panels
Separation	For leafs with more than one vision panel, enter the vertical offset between panels
Set Top Panel Height	For leafs with more than one vision panel, select to make the top vision panel a different height than the other panels and specify the desired **Panel Height**
Add Vision Panels to Unequal Leaf	For doors with an unequal leaf, select to include vision panels with the same parameters on the smaller leaf as well as the main leaf.
Glass	The Glass leaf type enables the options specific to glass doors on this pane, such as Muntin Style; it does not make the door leaves render as transparent glass. To make the glass render properly, apply a **Glazing** class on the Classes pane.
Thickness	Enter the thickness of the door slab
Top Rail Width	Enter the width of the top door rail
L/R Stile Width	Enter the width of the left and right stile
Bottom Rail Width	Enter the width of the bottom door rail
Muntin Style	Select the muntin style
Vertical/Horizontal Bars	Enter the number of vertical/horizontal muntin bars
Bar Width	Enter the muntin bar width
Bar Depth (Vectorworks Architect or Landmark required)	Enter the muntin bar depth
Bar Offset (Vectorworks Architect or Landmark required)	Enter the muntin bar offset for Prairie muntins
Panel	
Thickness	Enter the thickness of the door slab
Top Rail Width	Enter the width of the top door rail

Parameter	Description
L/R Stile Width	Enter the width of the left and right stile
Bottom Rail Width	Enter the width of the bottom door rail
Vertical/Horizontal Panels	Enter the number of vertical/horizontal panels in the door
Mid Stile Width	Enter the width of any interim stiles
Set Top Panel Height	Select to make the top door panel a different height than the other panels and specify the desired **Panel Height**
Single Top Panel (Vectorworks Architect or Landmark required)	Select to specify only one panel across the top of the leaf, regardless of the number of panels below
Glazing in Top Panels (Vectorworks Architect or Landmark required)	Select to insert glazing in the top panel(s)
Muntin Style	Select the muntin style
Vertical/Horizontal Bars	Enter the number of vertical/horizontal muntin bars
Bar Width/Depth	Enter the muntin bar width/depth
Bar Offset	Enter the muntin bar offset for Prairie muntins
Custom (Vectorworks Architect or Landmark required)	
Thickness	Enter the thickness of the door slab
Custom Leaf	Select the leaf type from the default content; see “Resource Libraries” on page 219. Custom leaves must be 3D-only symbols made from generic solids, saved in the default content location.

Door Settings: Lights Pane

Lights options are not enabled for the Opening **Configuration** or for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Include Sidelights	Select to include sidelights
Left/Right Width	Enter the left/right sidelight width
Sash Width	Enter the sidelight and transom sash width
Sash Depth	Enter the sidelight sash depth
Mullion Width	Enter the sidelight mullion width
Mullion Depth	Enter the sidelight mullion depth

Parameter	Description
Include Muntins	Select whether to add muntins to the sidelights
Style	Select the sidelight muntin style
Vertical/Horizontal Bars	Enter the number of vertical/horizontal muntin bars
Bar Width	Enter the sidelight muntin bar width
Bar Depth (Vectorworks Architect or Landmark required)	Enter the sidelight muntin bar depth
Bar Offset (Vectorworks Architect or Landmark required)	Enter the muntin bar offset for Prairie muntins

Door Settings: Threshold Pane

Vectorworks Architect or Landmark product required. Threshold options are not enabled for the Opening **Configuration.**

Click to show/hide the parameters.

Parameter	Description
Include Threshold	Select whether to include a threshold
(1) Depth (Int)/(2) Depth (Ext)	Enter the interior and exterior threshold depth
(3) Overall Depth	Enter the overall threshold depth
(4) Offset	Enter how far to offset the threshold from the center of the door jamb
(5) Thickness	Enter the threshold thickness
(6) Nosing	Enter the threshold nosing height
Threshold Under Leaf	Select to limit the threshold to under the leaf; otherwise, to extend the threshold beyond the inside of the jamb, enter the desired **Extension** width (this amount is applied to each side of the threshold, and negative values are not permitted)

Door Settings: Transom Pane

Transom options are not enabled for the Opening **Configuration** or for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Sash Width	Enter the transom sash width
Sash Depth	Enter the transom sash depth
Mullion Width	Enter the transom mullion width

Parameter	Description
Mullion Depth	Enter the transom mullion depth
Include Muntins	Select whether to add muntins to the transom
Style	Select the transom muntin style
Vertical/Horizontal Bars	Enter the number of vertical/horizontal muntin bars
Bar Width	Enter the transom muntin bar width
Bar Depth (Vectorworks Architect or Landmark required)	Enter the transom muntin bar depth
Bar Offset	Enter the muntin bar offset for Prairie muntins

Door Settings: Trim Pane

Trim options are not enabled for the Opening **Configuration** or for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Include Interior/ Exterior Trim	Select whether to include interior/exterior trim
Width/Depth	Enter the interior/exterior trim width and depth

Door Settings: Lintel Pane

Vectorworks Architect or Landmark product required. Lintel options are not enabled for the Opening **Configuration** or for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Include Lintel	Select to add a lintel above the door or transom
Int. / Ext. Protrusion	Enter the interior and exterior lintel protrusion
(1)Thickness	Enter the lintel thickness
(2)Angle	Enter the lintel angle
(3)Drop	Enter the length of the lintel drop

Door Settings: Hardware Pane

Hardware options are not enabled for the Opening **Configuration.**

Click to show/hide the parameters.

Parameter	Description
Include Hardware	Select to include a door hardware set with the door object; see “Assigning, Creating, Editing, and Deleting Door Hardware Sets” on page 629.
View Hardware	Click to open the View Door Hardware Set dialog box to view the parameters for the currently selected door hardware set; click **OK** to return to the Door Settings dialog box
Manage Hardware	Click to open the Door Hardware Library dialog box to assign, create, edit, or delete door hardware sets

Door Settings: Kick Plates Pane

Kick Plates options are not enabled for the Opening **Configuration** or for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Interior/Exterior Kick Plate	Select to include a kick plate on either side of the leaf
Height	Set the height of the kick plate While the height is variable, the kick plate’s distance from the right, left and bottom edges of the door is a fixed value of one inch/25.4 mm.
Class	Select a class to control visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the default door class to place the part in the same class as the door object.

Door Settings: Centerline Marker Pane

Vectorworks Architect or Landmark product required. Centerline markers options are not enabled for curtain wall doors.

Click to show/hide the parameters.

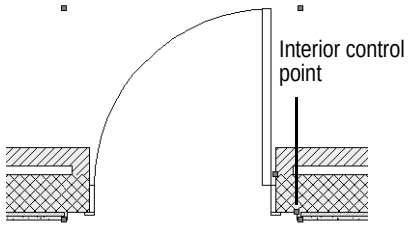
Parameter	Description
Centerline Marker	Select to display a centerline marker in Top/Plan view.
Size	Set the size of the centerline
Class	Select a class to control visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the default door class to place the part in the same class as the door object.
Line Attributes	Select the line attributes and weight, or select **Set Thickness** to open the Set Thickness dialog box for creating a custom line thickness. If **Use Class Attributes** is selected, the line attributes are set by the part’s class
Include Marker	Select whether to include a marker and choose the desired style from the marker list, or select **Custom** to create a custom marker. Select **Edit Marker List** to open the Edit Marker List dialog box; see “Editing the Marker List” on page 1103. If **Use Class Attributes** is selected, the marker attributes are set by the part’s class.
Use Class Attributes	Select to set the centerline marker’s attributes (including marker style) by class rather than the parameters in the Door Settings dialog box

Parameter	Description
Show Centerline Marker on Larger Leaf	For bipart swing doors with leafs of unequal size, select to center the line on the larger leaf instead of the opening

Door Settings: Interior Wall Detail Pane

Interior wall detail options are not enabled for curtain wall doors.

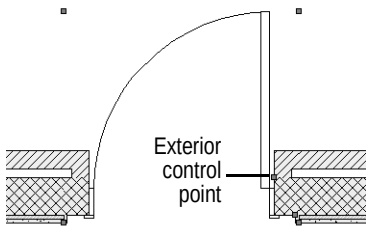
Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of interior wall components to wrap; when interior wall components are specified, the door enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the interior wall; the splay can be specified either as an angle/ diagonal or with width and depth values
(1) Angle	Enter the interior wall splay angle
(2) Diag	Enter the interior wall splay diagonal value
(3) Width	Enter the interior wall splay width
(4) Depth	Enter the interior wall splay depth

Door Settings: Exterior Wall Detail Pane

Exterior wall detail options are not enabled for curtain wall doors.

Click to show/hide the parameters.

Parameter	Description
Number of Components	Specify the number of exterior wall components to wrap; when exterior wall components are specified, the door enables a control point to set the wrapping point 
Splay Wall	Select whether to splay the exterior wall; the splay can be specified either as an angle/ diagonal or with width and depth values
(1) Angle	Enter the exterior wall splay angle

Parameter	Description
(2) Diagonal	Enter the exterior wall splay diagonal value
(3) Width	Enter the exterior wall splay width
(4) Depth	Enter the exterior wall splay depth

Door Settings: Classes Pane

The visibility of the overall 3D door is controlled by the **Class** setting on the Object Info palette; part settings are controlled from the Door Settings dialog box.

Click to show/hide the parameters.

Parameter	Description
Door parts	Select a class for each door part to control its visibility. The classes present in the drawing are listed; alternatively, create a new class. Select the door class to place the part in the same class as the door object. The Architect and Landmark products have a more comprehensive list of door parts available to be assigned a class. For doors with an Opening **Configuration**, in a 3D view a no fill/no pen weight extrude completely fills the wall in place of the door; this extrude is placed in the Interior Jamb class and is filtered out in IFC export operations. Change the class settings to control the visibility of the extrude, as needed

Door Settings: Data Pane

Certain data fields represent calculated values and cannot be edited; as a result, the **Field Name** and **Field Value** appear dimmed for those data fields.

Click to show/hide the parameters.

Parameter	Description
Field Name	Select the data field from the list and its name displays beneath the list; information associated with the **Field Name** can be specified in the Field Value area. The user fields can include additional user-defined information with the door.
Field Value	Enter data for use in the door schedule

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Doors
Inserting Doors in Vectorworks Fundamentals
Door Properties
Creating Door Schedules or Worksheets
Door Terminology Specific to the United Kingdom

Assigning, Creating, Editing, and Deleting Door Hardware Sets

A door hardware set can be assigned to the door, and a door operator device (such as a knob or lever) can be selected to display on the 3D door from default content.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To assign, create, edit, or delete a door hardware set:

1. From the Hardware pane of the Door Settings dialog box, select **Include Hardware** and click **Manage Hardware**.

   The Door Hardware Library dialog box opens for selecting a door hardware set and door operator device.

Click to show/hide the parameters.

Parameter	Description
Title Bar	Displays the name of the current preference set (see “Working with Project Preference Sets” on page 1866)
Door hardware set list	Lists all defined door hardware sets
Add	Creates a new hardware set
Remove	Removes the selected hardware set from the library (this action cannot be undone)
Edit	Edits the currently selected hardware set

If the Door Hardware Library.txt file (in [Vectorworks]\VW_Arch\Data\Prefs_Def) is already in use, the **Add** and **Remove** buttons on the Door Hardware Library dialog box are disabled and the **Edit** button changes to a **View** button. A Read Only heading will display in the title bar of the Door Hardware Library dialog box and the Edit Door Hardware Set dialog box is also disabled until the file is writable again.

2. Select **Edit** or **Add**.

   The Edit or Add Door Hardware Set dialog box opens. The Add dialog box is similar to the Edit dialog box except that all fields are blank.

Click to show/hide the parameters.

Parameter	Description
Description	Indicates the name of the door hardware set
Hdw Set ID	Indicates the unique ID value assigned to the door **HW Set** field in the door schedule
Hinge Qty	Indicates the number of hinges
Hinge Type	Indicates the hinge type
Lockset Type	Indicates the lockset type
Closer Type	Indicates the closer type
Stop Type	Indicates the stop type
Key Code	Indicates the key code
Notes	Contains any pertinent notes about the door operator device
Symbol	List the available door hardware to assign to the door from the default content

3. Complete each field with the desired information to add a new door hardware set, or edit the desired fields to change an existing door hardware set.
4. Click **OK**, and then click **Done** to return to the Door Settings dialog box.

   The applied information appears in the door hardware legend generated (see “Creating Schedules” on page 1864).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Door Settings: Hardware Pane
Creating Door Schedules or Worksheets

Door Properties

To edit door parameters in the Door Settings dialog box, click **Settings** from the Object Info palette, double-click the door to open the Door Settings dialog box, or right-click (Windows) or Ctrl-click (Mac) on a door and select **Edit** from the context menu.

When multiple door objects are selected for editing, and if the parameter settings of the selected objects are different, parameters display in an "indeterminate state." Any values changed through the dialog box are changed for all the selected objects.

Door objects can also be edited in the Object Info palette. If the door has been inserted as a plug-in object, most settings from the Door Settings dialog box display. If the door is a black symbol made from a door object, fields pertaining to door geometry do not display. See "Symbol Types" on page 237.

For the Vectorworks Architect and Landmark products, most, but not all, parameters can be edited from both the Door Settings dialog box and the Object Info palette for ease of access. The fields in the Object Info palette are named similarly (but not always identically) to those in the Door Settings dialog box, and roughly reflect the order in which settings are entered in the dialog box, for ease of editing. Curtain wall doors have limited parameter options.

For the Vectorworks Fundamentals and Spotlight products, certain frequently used parameters are available to be edited directly in the Object Info palette, and do not necessarily reflect the order in which settings are entered in the dialog box. All other parameters are edited only from the Door Settings dialog box.

The door parameters are described in "Inserting Doors in Vectorworks Fundamentals" on page 609 and "Inserting Doors in Vectorworks Design Series" on page 617. Only the parameters that are different are described here.

Click to show/hide the parameters.

Parameter	Description
Flip	Click to flip the orientation of doors inserted in a wall
Set Position	Click to activate the **Move by Points** tool in Reference Point mode and move the door by a specified distance from a reference point

Inserting Doors in Vectorworks Fundamentals
Inserting Doors in Vectorworks Design Series
Creating New Symbols
Moving Objects
Door Terminology Specific to the United Kingdom

Creating Door Schedules or Worksheets

In the Vectorworks Architect product, the door schedule and/or door hardware legend can be added to the drawing from the **VA Create Schedule** command (see "Creating Schedules" on page 1864) or the Resource Browser. From the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder that is included with the Vectorworks Architect product (see "Resource Libraries" on page 219). Drag the door schedule or door hardware legend worksheet to the drawing. The worksheet is populated with information from the objects in the current drawing. To edit the worksheet after it has been created, see "Using Worksheets" on page 1319.

Users of the Vectorworks Landmark and Spotlight products can also create the worksheet by creating a report for objects with a door record. To create the report from Landmark, select **Tools > Reports > Create Report**; from Spotlight, select **Spotlight > Reports > Create Report**.

Inserting Doors in Vectorworks Design Series
Door Properties
Moving Objects
Door Terminology Specific to the United Kingdom

Door Terminology Specific to the United Kingdom

Certain door terminology varies in the United Kingdom. The door object is automatically localized in the Vectorworks program for users with a United Kingdom serial number; however, since the documentation is not localized, the terminology does not match. This table shows how the terms are mapped.

U.S. Term (in Documentation)	U.K. Term (in Vectorworks Program)
Transom	Fanlight
Rough Opening	Structural Opening
Muntins (Leaf, Sidelight, Transom)	Glazing Bars
Colonial (Muntin Style)	Georgian
Mullion (Transom)	Transom Bar
Trim	Architraves
Interior Trim	Interior Architraves
Exterior Trim	Exterior Architraves
Masonry Opening (Data)	Nearest Full-Module of Masonry
Door Configurations	Door Configurations
Swing Simple	Single Leaf
Swing Bi-Part	Double Leaf
Pocket Simple	Pocket Single
Pocket Bi-Part	Pocket Bi-Parting
Bi-Fold Simple	Bi-Fold Single
Bi-Fold Bi-Part	Bi-Fold Double
Double Acting	Double-Swing Double-Leaf
Leaf Types	Leaf Types
Solid	Flush
Glass	Glazed

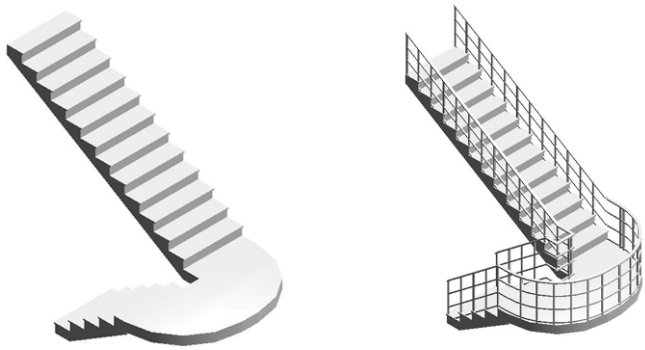
Inserting Doors in Vectorworks Fundamentals
Inserting Doors in Vectorworks Design Series

Stairs

Inserting Stairs

The Vectorworks Fundamentals, Landmark, and Spotlight workspaces include a **Simple Stair** tool for drawings that do not require a complex stair object. The Vectorworks Architect, Landmark, and Designer products include the **Stair** tool, and the workspaces can be edited to add the **Simple Stair** tool.

These tools allow you to build a stair with various geometries. The stair is a hybrid object, offering control over the 2D plan appearance, as well as the 3D component, for plan and presentation drawings. The **Stair** tool includes many options, details, and configurations, and performs many stair design chores automatically.



The **Stair** tool provides a set of common configurations (straight, L-shaped, U-shaped, O-shaped, and circular). It allows the creation of "styles" from sets of parameters controlling groups of geometry and display attributes, to be saved and recalled for use in other stair instances. The stair can be set to determine its own floor-to-floor height based on the layers the stair connects, and can take advantage of a project set up with stories. The stair can have separate lower floor and upper floor graphical representations. This avoids having to draw stairs on two different layers, and edit them separately, to completely represent the same physical staircase. Stair break options can be set differently on each layer, to get exactly the desired look in Top/Plan view. The entire stair configuration can be saved as a symbol, to be used again or shared with others, establishing a library of stair standards. Stair parameters can also be copied from one stair to another. While the geometric variation of the stair is restrained, the control over 2D and 3D graphic attributes is high. Stairs can be created as 2D only objects or as hybrid objects.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Simple Stair
Creating a Stair
Transferring Stair Properties

Creating a Simple Stair

A simple stair object is available to add a basic stair representation to the drawing in the Vectorworks Fundamentals, Landmark, and Spotlight workspaces. Vectorworks Architect, Landmark, and Designer products contain a more advanced stair tool for designs that require complex, detailed stairs (see "Inserting Stairs" on page 633).

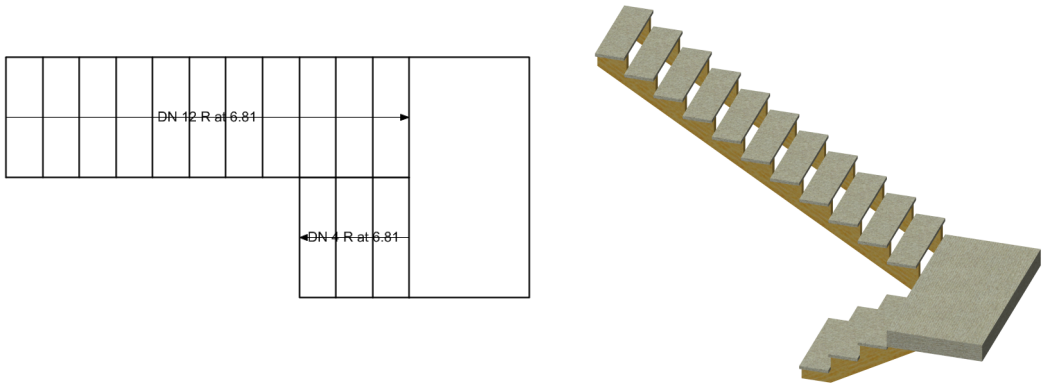
To insert a simple stair:

1. Click the **Simple Stair** tool from the appropriate tool set:
   - Fundamentals workspace: Wall tool set
   - Landmark and Spotlight workspaces: Building Shell tool set
2. Click to place the simple stair in the drawing, and click again to set the stair's rotation. If this is the first time a stair is placed in the drawing, the stair object properties dialog box opens. Set the parameters, which apply to subsequently created stairs.

   The stair parameters can also be edited from the Object Info palette.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Click to show/hide the parameters.

Parameter	Description
Style	Select the stair style: standard, indented, open riser, or masonry
Width	Specifies the width of the staircase
Flr-Flr Height	Specifies the height of the staircase from floor to floor
Max Riser	Indicates the horizontal maximum thickness of the riser
Tread Depth/Height	Sets the tread depth (front to back) and the tread height
Nosing Depth	Specifies the nosing depth
Config	Select the stair configuration: straight, straight with a landing, left and right landings, and left and right double-back stairs with landings (even and irregular)
Landing Tread	Indicates how many steps occur before the landing (does not apply to even double back stairs)
Separation	For double back style stairs, sets the separation distance between the flights
Offset 1/2	For stairs with landings, indicates the distance to offset the landing from the stairs on each side of the landing
Curved Landing	Creates curved landings; the radius matches the stair width
Stringer Width/Offset	Sets the stringer width (thickness) on each side of the stair, and specifies the distance between the outside of the stair tread to the outside of the stringers
Draw Lower Stair Break	Hides a portion of the stair with a stair break, so that the lower portion of the stair can be displayed in 2D
Show Data	Select to display the number of risers and riser height along with the path-of-travel arrow; an Up or Down arrow must be selected in **Arrows** for stair data to be displayed
Draw Top Tread	Specify whether the stair should include a tread at the same level as the upper floor it serves
Create 3D	Creates a 3D version of the stair to display in views other than Top/Plan
Left/Right Rail	Adds handrails to the left and right of the staircase
Rail Height/Width	Sets the height and width for the railing(s)
Arrows	Select whether to display the path-of-travel arrow, and in which direction
Step/Rail/Stringer Finish	Select the **Class** to assign to the steps, railing(s), and stringers



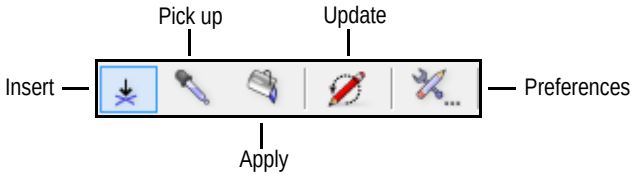
A L Creating a Stair

The **Stair** tool is available in the Vectorworks Architect, Landmark, and Designer workspaces. If the stair will have a lower layer and upper layer representation, or if its total rise will be defined by layer or story elements, it should be inserted on the layer representing the lower of the two floors it connects.

To insert a stair:

1. Click the **Stair** tool from the Building Shell tool set.

   Five modes are available.



Mode	Description
Insert	Inserts a stair with the parameters specified in Preferences
Pick up	Picks up selected attributes from a stair to be transferred to another stair; attributes are selected in the stair preferences
Apply	Transfers selected attributes from one stair to another; attributes are selected in the stair preferences and picked up in Pick up mode.
Update	Updates all stairs in the drawing, which is necessary when layer scale has changed and a change to the stair's schematic representation is also required
Preferences	Opens the Stair Settings dialog box, to set default stair parameters

2. Select **Insert** mode and then click **Preferences** from the Tool bar before placing the stair to set the default stair parameters. Click **OK** to set the stair parameters and close the Stair Settings dialog box.

   Alternatively, insert the default stair and then click **Settings** from the Object Info palette, or simply double-click on an existing stair, to set the stair parameters.

3. When the stair configuration parameters on each tab are set, consider whether to save the stair as a symbol.

If the same customized stair is to be used numerous times in a drawing, or will be used in another drawing or by another designer, this eliminates the need to repeatedly apply parameters, maximizes memory efficiency, and allows global editing of symbols. See “Creating New Symbols” on page 239.

If you don’t wish to create a symbol, the stair attributes can easily be transferred.

4. Click in the drawing to insert the stair. Click again to set the stair’s rotation.

To edit stair parameters for a placed stair, click **Settings** from the Object Info palette, double-click the stair to open the Stair Settings dialog box, or right-click (Windows) or Ctrl-click (Mac) on a stair and select **Edit** from the context menu. Depending on the stair settings, certain parameters are available directly from the Object Info palette.

Each tab of the stair parameters is described in the following sections. As the parameters are defined, the preview dynamically displays the stair appearance.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Stair Properties
Transferring Stair Properties
Inserting Stairs

A L Stair Settings: General Tab

1. Click the General tab to set the stair configuration.

   Select a predefined configuration as the basis for the new stair. The general configuration can be based on either pre-defined default content symbols, custom default files in the user folder, a saved symbol in the current file, or by selecting a typical stair configuration.

   - From the **Symbol Selection** list, select an initial stair configuration from the current file, custom defaults, or from the standard configurations. Once a selection has been made, a symbol selection list allows the symbol or predefined symbol to be selected from the file, custom, or default content (see “Resource Libraries” on page 219).
   - Click **Select Standard Configuration** to select an initial stair configuration that most closely matches the type of stair to create. The Select a Stair Configuration dialog box opens; select one of the configurations. If there are any current stair parameters that should be saved and transferred to the new stair, select the category or categories from **Transfer settings from current stair**.

   When the stair configuration has been selected, the available parameters on the tabs change to reflect the selection.

2. Once the initial configuration has been selected, change parameters as necessary to define the stair.

   Minimum and maximum value ranges can be set for the stair angle, tread depth, riser height, and step length.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
General Settings	
Component	Select whether to create a 2D only or hybrid 2D/3D object
Class	Select a class for the stair. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the currently active class, or select a default class name.
Orientation	For stairs with oblique/winder treads, specifies the side of the stair with the shortest tread depth; for L- and U-Stairs, determines the direction of travel
General Geometry	Specify overall parameters for the stair configuration. These general parameters can be set here or on the Geometry tab. Values entered here are also set on the Geometry tab. For stairs with multiple flights, the **Stair Width** value entered on the General tab is applied to all flights on the Geometry tab.
Lock button	For any parameter, locks the parameter value, forcing the other parameters to recalculate to automatically adjust
Preview	Displays a preview of the current stair configuration
Preview for	Select either a detailed or schematic view for the preview
Stair detail status (in parentheses)	Stair details may be hidden at certain layer scales, depending on the detail display setting on the Display tab of the document preferences (see “Document Display Preferences” on page 60). Depending on the setting, the status indicates whether stair details will display for stairs in the drawing, or shows the scale required for the details to display.
View	Select a view for the preview from the list of standard views; 3D views are not available for a 2D only stair
Render Mode	Select a render mode for the preview from the list of available modes
Calculate	Forces the stair object the recalculate based on the current set of parameter values
Previous Stair	Loads the parameter values of the most recently created stair
Save as Symbol	Creates a symbol based on the current settings
Use Minimum/ Maximum Values	Enables the use of minimum/maximum settings. Normally, this option should be enabled to create stairs with valid proportions and settings that meet standard requirements. However, during the initial design phase, it may be convenient to temporarily disable the settings while exploring various geometry and design possibilities.
Edit	Opens the Min./Max. Settings dialog box

3. If you switch to another stair configuration after making parameter changes on any tab, the New Stair dialog box opens.

   Switching to a new configuration loses any parameter settings that have already been made. The settings from the current stair configuration can be transferred to the new stair if needed. Select the settings to transfer, or click **All** to select all settings and **None** to deselect all the settings.

Stair Settings: Geometry Tab

Stair Settings: 2D Graphics Tab
Stair Settings: Construction Tab
Stair Settings: Railings Tab
Stair Settings: Graphic Attributes Tab
Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Stair Properties
Transferring Stair Properties
Creating a Stair
Inserting Stairs

A L Stair Settings: Geometry Tab

1. Click the Geometry tab to set both basic and detailed stair geometry based on the configuration selected on the General tab. The geometry parameters automatically influence each other; entering a few desired parameters causes the remaining parameters to adjust accordingly.

   The diagrams next to each parameter indicate the relevant area of the stair.

Click to show/hide the parameters.

Parameter	Description	Applies to
Lock button 	For any parameter, locks the parameter value, forcing the other parameters to recalculate to automatically adjust	All
Link button 	Links stair flight widths so that they are set to the same value automatically; enter a value for one of the widths, and click **Link** to enter the same value for all linked flights	Stairs with multiple flights
Total Rise: By Value or By Layer Elevation	Sets the floor-to-floor height of the stair by either a value or by layer elevation settings of the layers/levels it connects. If by layer elevation, the Total Rise By Layer Elevations dialog box opens; see Step 2.	All
Tread Depth(G)	Specifies the average depth of the stair tread, measured from the face of the riser to the face of the next riser. Also known as the “going.”	All
Riser Height(R)	Indicates the average height of the stair risers	All
Step Length(2R+G)	Displays the step length, which is calculated as 2 * the Riser Height (R) + Tread Depth (G)	All
Stair Width	Sets the overall width of the stair	Stairs with a single flight
Stair Width 1/2/3/4/5	Sets the width of each flight of stairs	Stairs with multiple flights L-Stair, Single Landing L-Stair with winders U-Stair, Single/Double Landings U-Stair with winders Double-U Stair, Double Landings Triple-L Stair, Triple Landings

Parameter	Description	Applies to
Length	Specifies the total length of the stair, measured from the nosing of the starting tread to the end of the last tread	Straight Stairs (all) L-Stair, Single Landing L-Stair with winders (all) U-Stair, Single/Double Landings U-Stair with winders (all) Circular Stair
Walk Line Length	Indicates the length of the line that determines the direction of the stair and is the critical line from which the tread depth, oblique dimensions, and other values are calculated	Straight Stairs (all) L-Stair, Single Landing L-Stair with winders (all) U-Stair, Single/Double Landings U-Stair with winders (all) Circular Stair
Draw Top Tread	When enabled, the top tread is drawn; its parameters can be specified independently from the rest of the stair. Select the parameter from the list, and enter its value.	All
Min. Inside Tread Depth	Specifies the minimum allowable tread depth value for oblique, winder, or fixed angle winder stair treads	All
Walk Line Offset Options	Sets the walk line location • **Walk Line by Offset:** Sets the distance to offset the walk line from the inside of the stair; enter the offset value • **Walk Line on Center:** For stairs with fewer than two stair widths, sets the walk line offset to 0.5 times the stair width • **Walk Line Centered on All Flights:** For stairs with several flights, sets the walk line offset to 0.5 times the stair width, centering the walk line for all flights. If the stair has different flight widths, the walk line displays centered for all flights, but the calculation of treads and winders is based on the first flight only. • **Walk Line Centered on First Flight:** For stairs with several flights, sets the walk line offset to 0.5 times the stair width. For stairs with winders, the walk line is centered based on the first flight only.	All
Oblique Bottom Tread	Offsets the distance of the front edge of the first tread from the start of the overall length of the stair (as measured along the walk line)	All
Oblique Top Tread	Offsets the distance of the front edge of the last tread from the start of the overall length of the stair (as measured along the walk line)	All

Parameter	Description	Applies to
Top/Bottom Treads	Opens the Top/Bottom Treads dialog box to access certain parameters: • **Draw Top Tread:** Select to draw the top tread and set the parameters independently from the rest of the stair • **Minimum Inside Tread Depth:** Specifies the minimum allowable tread depth value for oblique, winder, or fixed angle winder stair treads • **Oblique Bottom Tread:** Offsets the distance of the front edge of the first tread from the start of the overall length of the stair (as measured along the walk line) • **Oblique Top Tread:** Offsets the distance of the front edge of the last tread from the start of the overall length of the stair (as measured along the walk line)	Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Num. of Risers	Sets the number of risers for the stair	Straight, Single Flight Run L-Stair, Winder U-Stair, Single/Double Winder Return Circular Stair
Num. of Risers 1	Sets the number of risers for the first or bottom flight of stairs	Straight, Single/Double Landings L-Stair, Single Landing U-Stair, Single/Double Landings Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Num. of Risers 2	Sets the number of risers for the second or middle flight of stairs	Straight, Single/Double Landings L-Stair, Single Landing U-Stair, Single/Double Landings Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Num. of Risers 3	Sets the number of risers for the third flight of stairs	Straight, Double Landing U-Stair, Double Landings Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Num. of Risers 4	Sets the number of risers for the fourth or top flight of stairs	Triple-L Stair, Triple Landings
Landing Length 1	Indicates the landing length	Straight, Single/Double Landings L-Stair, Single Landing U-Stair, Single/Double Landings
Landing Length 2	Sets the length of the second landing	Straight, Double Landing U-Stair, Double Landing
Side 1	Indicates the overall outside length of the stair, as measured from the start or bottom of the stair to the outside corner of the landing	L-Stair, Single Landing U-Stair, Single/Double Landings Double-U Stair, Double Landings Triple-L Stair, Triple Landings

Parameter	Description	Applies to
Side 2	Indicates the overall outside length of the stair, as measured from the outside corner of the landing to the end or top of the stair, or to the outside corner of the next landing	L-Stair, Single Landing U-Stair, Single/Double Landings Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Side 3	Indicates the overall outside length of the stair, as measured from the outside corner of the landing to the end or top of the stair, or to the outside corner of the next landing	U-Stair, Single/Double Landings Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Side 4	Indicates the overall outside length of the stair, as measured from the outside corner of the landing to the end or top of the stair, or to the outside corner of the next landing	Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Side 5	Indicates the overall outside length of the stair, as measured from the outside corner of the landing to the end or top of the stair	Double-U Stair, Double Landings
Offset 1	Specifies the distance from the front edge of the landing or last riser of the first flight to the inside corner of the first landing	L-Stair, Single Landing U-Stair, Single/Double Landings, Fixed Angle Winder stairs Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Offset 2	Specifies the distance from the inside corner of the landing to the back edge of the landing or first riser of the next flight	U-Stair, Double Landings, Fixed Angle Winder stairs Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Offset 3	Specifies the distance from the front edge of the landing or last riser of the second flight to the inside corner of the second landing	U-Stair, Double Landings, Double Fixed Angle Winder Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Offset 4	Specifies the distance from the inside corner of the second landing to the back edge of the second landing or first riser of the next flight	U-Stair, Double Landings, Double Fixed Angle Winder Double-U Stair, Double Landings Triple-L Stair, Triple Landings
Offset 5	Specifies the distance from the front edge of the landing or last riser of the third flight to the inside corner of the third landing	Triple-L Stair, Triple Landings
Offset 6	Specifies the distance from the inside corner of the third landing to the back edge of the third landing or first riser of the next flight	Triple-L Stair, Triple Landings
Outside Diameter	Sets the diameter measured from the outside of the stair	Circular Stair
Inside Diameter	Sets the diameter measured from the inside of the stair	Circular Stair
Sweep	Sets the angle of the arc formed by the walk line	Circular Stair

2. If setting the stair **Total Rise** by layer elevation instead of by a specific value, the height of the stair can be constrained by layer or story elements at its top and bottom boundaries; the lower floor and upper floor indicate the two stories the stair connects. Adjusting a bounding element causes the height of the stair to automatically change accordingly.

Select **By Layer Elevation**; the Total Rise by Layer Elevations dialog box opens.

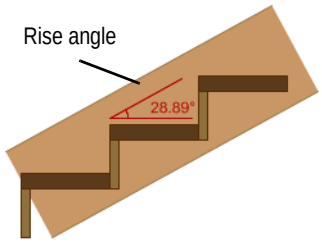
Click to show/hide the parameters.

Parameter	Description
Top Bound	Sets the vertical reference that determines the top of the stair. The **Layer Wall Height** value is set by the design layer (see "Setting Design Layer Properties" on page 165). Alternatively, the top of the stair can be bound by one of the story levels defined for the story or the story above it. By setting the top of the stair to a level type, if the elevation of the associated story changes, the height of the stair changes automatically to match.
Top Offset	Sets the offset of the top of the stair from its specified top bound height
Bottom Bound	Sets the vertical reference that determines the bottom of the stair. Alternatively, the bottom of the stair can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the stair to a level type, if the elevation of the associated story changes, the height of the stair changes automatically to match.
Bottom Offset	For the bottom of the stair, sets the offset from its specified bottom bound height
Z of Top Bound	Displays the elevation value of the stair's upper boundary
Z of Bottom Bound	Displays the elevation value of the stair's lower boundary
Calculated Total Rise	Displays the calculated total rise value for the stair based on the stair's upper and lower boundaries

Click **OK** to set the stair total rise.

3. The Preview contains two preview graphics.

- The lower preview graphic displays a detailed section/elevation view of the treads and risers, and indicates the current rise angle.



- The upper preview graphic displays the Top/Plan view and is interactive.
  - Click on an upper or lower tread to set the oblique tread parameters for the highlighted treads.
  - L- and U-Stairs have clickable red areas at the corner of the landing. Click each area to set corner parameters for the inside and outside corners of the landing. The Stair Landing Corners dialog box opens. For configurations that allow angles other than 90 degrees, enter the desired **Angle between flights**; the corner style parameters update automatically. For each corner, set the **Corner Style** to the desired angle, or round the corner by specifying a chamfer or fillet value.

- For Double-U and Triple-L configurations, some parts of the stair may overlap in the Top/Plan preview, making it difficult to select and edit the intended corner parameters. Before clicking the red areas to set corner parameters for these configurations, select whether the preview displays the **Upper part of stair** or **Lower part of stair**.
- For fixed angle winder stair configurations, click on the stair winder to set the winder parameters. The Winder Parameters dialog box opens. Specify the number of treads for the winder section of the stair, and select whether the winder tread depths should match the tread depths for the straight parts of the stair. If maintaining a constant tread depth is desired, the minimum winder tread depth may need to be adjusted to prevent errors.

Stair Geometry Errors

The Geometry tab allows values to be locked, which forces other parameters to recalculate while maintaining appropriate calculated values for the stair. Occasionally, a invalid entry is attempted, which generates an error due to other locked values. An alert dialog box notifies you of the error and offers several possible options to avoid it.

Situations which can generate an alert include:

- The total rise calculation is based on multiplying the number of risers by the riser height, and therefore cannot be edited without changing one of the values. Similarly, the number of risers or riser height cannot be edited when the total rise is locked.

Parameter to be edited	Locked	Unlocked	Resolution
Total rise	Riser Height	Num. of Risers	Change the riser height or the total rise and number of risers as suggested, or return to previous values
Total rise	Num. of Risers	Riser Height	Modify the riser height, or the riser height and the number of risers as suggested, or return to previous values
Total rise	Riser Height and Num. of Risers	N/A	Change the riser height and/or the number of risers as suggested, or return to previous values
Riser height	Total rise	N/A	Modify the total rise, or the riser height and number of risers as suggested, or return to previous values
Number of risers	Total rise	N/A	Change the total rise, or the riser height/step length, or the riser height and number of risers as suggested, or return to previous values

- The total rise of the stair and the stair length (or sweep angle, for circular stairs) impact other values such as tread depth, riser height, and the number of risers. When the length/sweep angle is locked, it may become impossible to obtain valid entries for the other parameters. Since the stair length or sweep angle is a multiple of the tread depth, one cannot be edited when the other is locked.

Parameter to be edited	Locked	Resolution
Total rise	Length or Sweep Angle	Change the number of risers and step length, or unlock the length/ sweep angle as suggested, or return to previous values
Tread depth	Length or Sweep Angle	Modify the length/sweep angle, and/or change the number of risers and the tread depth as suggested, or return to previous values
Length or Sweep Angle	Tread Depth	Change the tread depth, and/or the number of risers, or change the risers and length/sweep angle as suggested, or return to previous values

Stair Settings: General Tab
Stair Settings: 2D Graphics Tab
Stair Settings: Construction Tab
Stair Settings: Railings Tab
Stair Settings: Graphic Attributes Tab
Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Stair Properties
Transferring Stair Properties
Creating a Stair
Inserting Stairs
Setting Up the Building Structure with Stories

Stair Settings: 2D Graphics Tab

The remaining four stair configuration tabs have a method of saving and recalling each tab's settings or style. The settings on the tab can be saved as a set and then loaded, replacing the current tab parameters.

1. Click the 2D Graphics tab to set the stair's 2D appearance on lower and, if desired, upper floors. The lower floor is always the floor where the stair has been placed. Selecting display options for the upper floor is optional; the upper floor representation is usually on another layer, and can be specifically selected here. These parameters apply only to the Top/Plan view, and do not affect the 3D appearance of the stair.

Click to show/hide the parameters.

Parameter	Description
2D Graphic Styles	Saves and recalls styles from a saved set
View	Determines the view to apply the parameters to the stair; depending on drawing settings like layer scale and the document preference **Hide details**, the detailed or schematic view status is displayed
Detailed	Applies the 2D graphic parameters to stairs in a detailed layer scale view
Schematic	Applies the 2D graphic parameters to stairs in a schematic scaled view
Accept for all Views	Applies the 2D graphic parameters to stairs in all views (detailed and schematic)
Stair Break Settings	Sets the stair break parameters; select the stair break configuration, and then refine the display with the additional parameters
Break Elevation	Sets the Z-height (relative to Layer Z=0) for the stair break
Break Line Angle	Specifies the angle of the break line, measured from the leading edge of the tread/riser
Break Lines Separation	Sets the distance between the pair of parallel break lines, measured in page scale units
Break Line Extensions	Indicates how far to extend the break lines past the sides of the stair
Draw Outlines at Break	When selected, draws boundary outlines around the stair object

Parameter	Description
Show Markers at Break	When selected, adds start and end markers to the walk line above and below the stair break
End of Walk Line is:	Sets the end of the walk line at either the last tread before the stair break or directly at the break
2D Graphics Settings	Controls the display of the walk line, nosing, stringers, headroom outline, and railings in Top/Plan view only
Show Nosing/Riser Offset	Displays the stair nosing and riser offset
Show Tread Offset	Displays the left and right tread offset
Show Stringers	Displays the stair stringers
Show Headroom Outline	Displays the headroom outline; specify the headroom height above the stair
Show Handrail	Displays the handrail (railing parameters are set on the Railings tab)
Show Top Rail	Displays the top rail of the handrail and/or guardrail
Show Posts	Displays railing posts
Show Railing above Stair Break	Displays the railing above the stair break
Lower Floor	The stair always displays on the lower floor (the floor where it has been placed); set the display of stair breaks, walk lines, arrow directions, and stair text
Display Stair	Select the stair display configuration for the lower floor • **Plain (no Break):** Does not draw a stair break; the stair is drawn as set by the 2D attributes on the Graphic Attributes tab • **Solid with Break:** Draws a stair break; the stair is drawn as set by the 2D attributes and 2D Stair Break on the Graphic Attributes tab, but line styles remain solid • **Below Break Only:** Draws a stair break; all geometry above the stair break is hidden • **Above Break Only:** Draws a stair break; all geometry below the stair break is hidden • **Break, Dashed Above:** Draws a stair break; all line styles above the stair break are automatically dashed according to the settings on the Graphics Attributes tab • **Break, Dashed Below:** Draws a stair break; all line styles below the stair break are automatically dashed according to the settings on the Graphics Attributes tab • **Dashed Above Only:** Draws a stair break; all line styles above the stair break are automatically dashed according to the settings on the Graphics Attributes tab • **Dashed Below Only**: Draws a stair break; all line styles below the stair break are automatically dashed according to the settings on the Graphics Attributes tab
Walk Line Below/ Above Stair Break	When selected, draws a walk line below/above the stair break. For Plain (no break) **Display Stair** style, draws a walk line the full length of the stair. When a walk line is indicated, select whether to display a start and/or end marker on the ends of the walk line (to show markers at a break, **Show Markers at Break** must be enabled). Set the arrow direction. The marker style is set on the Graphic Attributes tab.
Draw Entire Break	For Below Break Only and Dashed Below Only, select whether to draw the top of the stair break line, in addition to the bottom of the stair break line (which is always shown)

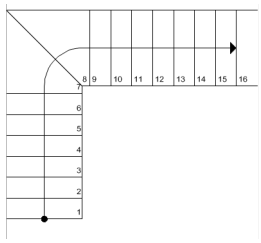
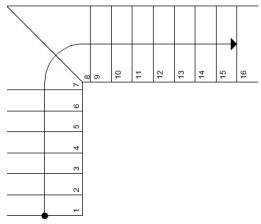
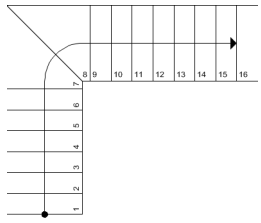
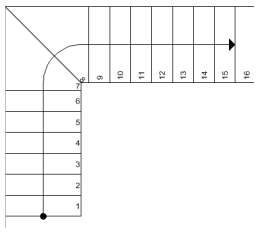
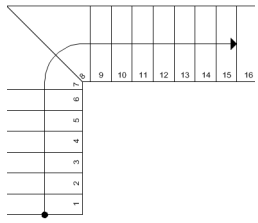
Parameter	Description
Stair Data	Opens the Stair Data Settings dialog box, for setting the appearance of stair text on the lower floor (see Step 2)
Upper Floor	Select to display the stair on the upper floor in both detailed and schematic views, and set the display of stair breaks, walk lines, arrow directions, and stair text for the upper floor
Layer	Specify the layer where the upper floor representation of the stair will be displayed. This selection affects both the detailed and schematic display of the stair on the upper floor. It is not related to the total rise stair height selection on the General or Geometry tab.
Upper floor stair configuration	The Display Stair, walk line, and Stair Data options function the same as for the lower floor
Break Offset	Adjusts the position of the stair break along the walk line. Enter a positive or negative offset to position the break without any gaps between upper and lower floors.
Preview	Displays a preview of the upper and lower floors of the 2D stair with the parameters applied

2. Click **Stair Data** for the lower, and optionally, the upper floor, to set the display of the 2D stair information text.

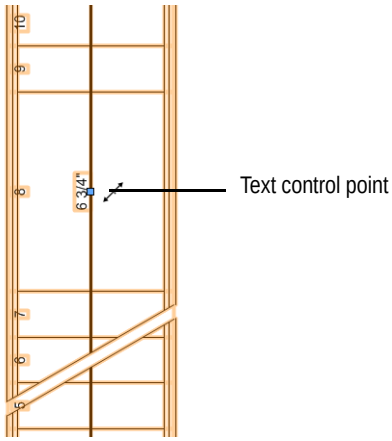
   The Stair Data Settings dialog box opens. The stair configuration determines the number of available run or flight parameters. Stair data text appearance is set on the Graphic Attributes tab.

Click to show/hide the parameters.

Parameter	Description
Show Numbers of Risers	When selected, displays the riser numbers for the stair. Specify the text position and offset for the flight and the step, and set the text rotation.
Position in flight	For straight stairs, indicate whether riser numbers display on the center, left, or right of the flight. For other stairs, indicate whether riser numbers display centered, or on the inner or outer side of the stair flight. The **Offset** value sets the distance between the riser numbers and the stair boundary.
Position in step	Sets the riser position centered on the step or at the start of the tread. The **Offset** value sets the distance between the riser numbers and the start of the tread, for numbers at the start of the tread only.

Parameter	Description
Rotation	Specifies the rotation of the riser numbers  **Horizontal in stair**: riser numbers always remain horizontal  **Vertical in stair**: riser numbers always remain vertical  **Aligned to edge**: riser numbers rotate by flight angle  **Aligned to tread**: riser numbers rotate with tread angle  **Perpendicular to tread**: riser numbers remain perpendicular to tread edge
Run	For each available run of stairs with multiple flights, select whether to include text for the total rise, average riser height, average tread depth, arrow direction (Up or Down, depending on walk line direction), and/or walk line length. As each item is selected for display, its code appears in the text box; text can also be typed between items. Select Line Break to add a carriage return between items. Set the angle of the text.
Note	For the entire stair, select whether to include text for the total rise, average riser height, average tread depth, arrow direction (Up or Down, depending on walk line direction), and/or walk line length. As each item is selected for display, its code appears in the text box; text can also be typed between items. Select Line Break to add a carriage return between items. Set the angle of the text.

The position of the stair run and note text can be adjusted by moving its control point.



3. Click **OK** to return to the Stair Settings dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Stair Settings: General Tab
Stair Settings: Geometry Tab
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Stair Settings: Construction Tab
Stair Settings: Railings Tab
Stair Settings: Graphic Attributes Tab
Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Stair Properties
Transferring Stair Properties
Creating a Stair
Inserting Stairs

A L Stair Settings: Construction Tab

The settings on the tab can be saved as a set and then loaded, replacing the current tab parameters.

Click the Construction tab to specify stair construction parameters. The parameters displayed depend on the selected construction configuration.

Click to show/hide the parameters.

Parameter	Description	Applies to
Construction Styles	Saves and recalls styles from a saved set	All
View	Determines the view to apply the parameters to the stair; depending on drawing settings like layer scale and the document preference **Hide details**, the detailed or schematic view status is displayed	All
Detailed	Applies the construction parameters to stairs in a detailed layer scale view	All
Schematic	Applies the construction parameters to stairs in a schematic scaled view	All
Accept for all Views	Applies the construction parameters to stairs in all views (detailed and schematic)	All
Construction Configuration	Select the type of construction for the stair	All
Solid Stair	Models the stair structure as single, monolithic geometry, but still includes treads and risers	All
Stringer Underneath	Models the stair structure as one, two, or three stringers (carriages) underneath that support the treads and risers above them. If three stringers are specified, the third will automatically be centered between the others, not the edge of the steps.	All
Stringer Outside	Models the stair structure as two stringers (carriages) on either side of the stair which support the treads and risers above them	All
Concrete Stair	Models the stair structure as monolithic, without separate treads or risers	All
Circular Stair	For circular stair configurations, select to obtain circular stair construction parameters	All

Parameter	Description	Applies to
Construction Settings		All
Upper Floor Thickness	Wraps the 3D stair geometry at the top of the stair around the edge of the floor slab by the indicated distance, correctly terminating all parts related to the construction type	All
Lower Floor Thickness	Wraps the 3D stair geometry at the bottom of the stair around the edge of the floor slab by the indicated distance, correctly terminating all parts related to the construction type. If set to 0 (zero) truncates the stair geometry so the bottom of the stair rests on top of the slab	All
Landing Thickness	Specifies the thickness of the stair landing, excluding any treads. The landing thickness plus the tread thickness is equal to the distance from the top of the tread to the bottom of the landing (solid stairs) or stringer (stringer underneath)	Stairs with a single landing
Landing Thickness 1	Specifies the thickness of the first stair landing, excluding any treads	Stairs with two landings
Landing Thickness 2	Specifies the thickness of the second stair landing, excluding any treads	Stairs with two landings
Solid Structure Thickness	When selected, draws 3D geometry below the treads and risers representing the structural mass of the stair. Specify the distance from the rear face to the inside edge where the riser and tread meet.	Solid Concrete
Tread Thickness	When selected, draws 3D stair tread geometry; specify the tread thickness	Solid Stringer Underneath Stringer Outside
Riser Thickness	When selected, draws 3D stair riser geometry; specify the riser thickness	Solid Stringer Underneath Stringer Outside
Nosing/Riser Offset	When selected, draws both 2D and 3D stair nosing geometry; set the distance that the nosing extends past the riser face over the preceding tread	All
Create Top Riser and Nosing	For stairs with the **Draw Top Tread** option disabled on the Geometry tab, select whether to display the top riser and tread nosing in 3D views	All
Offset Right (or Inner Offset)	Offsets the stair body from the right/inner edge of the stair treads and riser by the distance specified; as determined by the stair width, configuration, and orientation	Solid Stringer Underneath
Offset Left (or Outer Offset)	Offsets the stair body from the left/outer edge of the stair treads and riser by the distance specified; as determined by the stair width, configuration, and orientation	Solid Stringer Underneath
Stringer Depth	Sets the stringer's structural clear depth	Stringer Underneath Stringer Outside
Stringer Width	Sets the thickness (width) of the stringer	Stringer Underneath Stringer Outside

Parameter	Description	Applies to
Stringer Distance	For stairs with one stringer, specifies the stringer's offset distance from the center of the stair	Stringer Underneath
Tread/Stringer Offset	Sets the tread's offset distance from the bottom of the stringer's depth, as measured to the back, bottom edge of the tread	Stringer Outside
Stringer Extension at Stair Bottom	Specifies the distance that the stringer extends past the front edge of the starting (bottom) tread	Stringer Outside
Stringer Extension at Stair Top	Specifies the distance that the stringer extends past the front edge of the last (top) tread	Stringer Outside
Column	When selected, adds a central column	All, when stair configuration is Circular Stair
Custom Column Height	Specifies the central column height for spiral stairs	All, when stair configuration is Circular Stair
Custom Column Diameter	Specifies the central column diameter for spiral stairs	All, when stair configuration is Circular Stair
Outside Nosing	Sets the value for the tread nosing at the outside diameter of a circular stair	Circular stair, when stair configuration is also circular stair

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Stair Settings: General Tab
Stair Settings: Geometry Tab
Stair Settings: 2D Graphics Tab
Stair Settings: Railings Tab
Stair Settings: Graphic Attributes Tab
Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Stair Properties
Transferring Stair Properties
Creating a Stair
Inserting Stairs

A L Stair Settings: Railings Tab

The settings on the tab can be saved as a set and then loaded, replacing the current tab parameters.

Click the Railings tab to set the stair's handrail and guardrail parameters. The Railings tab is divided into three panes. On the left, the handrail and guardrail categories display, along with each current setting for the left, right, or both sides of the stair. Handrail and guardrail parameters are set separately. Click a disclosure arrow to reveal handrail or guardrail parameter categories. Select a category to edit its parameter settings in the middle pane. On the right, three previews display the railing cross-section, top/plan stair view, and isometric stair view.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Railing Styles	Saves and recalls styles from a saved set
View	Determines the view to apply the parameters to the stair; depending on drawing settings like layer scale and the document preference **Hide details**, the detailed or schematic view status is displayed
Detailed	Applies the railing parameters to stairs in a detailed layer scale view
Schematic	Applies the railing parameters to stairs in a schematic scaled view
Accept for all Views	Applies the railing parameters to stairs in all views (detailed and schematic)
Accept Settings for	Select the railing to which the current settings apply
Right/Left or Inside/Outside Railing	Applies the parameters to the selected side only
Both Sides	Applies the parameters to railings on both sides
Handrail	
Show Handrail	When selected, adds a handrail to the stair and enables the handrail parameters; **Show** displays for the handrail settings
Position	
Height	Sets the vertical distance from the leading edge of the tread/nosing to the top of the handrail
Handrail Position	Select whether the railing is on the stair or next to the stair
Offset to Tread Edge	Indicates the distance between the edge of the stair (the solid structure or the stringers) and the handrail
Top Rail	
Profile Type	Select the basic geometry of the top rail • **Line:** A 3D polyline serves as the railing profile • **Flat:** A 3D polygon with width, but no height, serves as the railing profile • **Round:** The cross-section is round, controlled by width • **Rectangular:** The cross-section is rectangular, controlled by width and height
Width	Sets the width for flat, rough, and square top railing geometry
Height	Sets the height of rectangular top railing geometry
Bottom Extension	Specifies the distance the top rail extends past the leading edge of the first tread or start of the stair; select whether the bottom extension is horizontal to the bottom riser or slopes with the stair
Top Extension	Specifies the distance the top rail extends past the leading edge of the last/top tread/end of the stair; select whether the top extension is horizontal to the bottom riser or slopes with the stair
Guardrail	

Parameter	Description
Show Guardrail	When selected, adds a guardrail to the stair and enables the guardrail parameters; **Show** displays for the guardrail settings
Position	
Height	Sets the vertical distance from the leading edge of the tread/nosing to the top of the guardrail
Guardrail Position	Select whether the railing is on the stair or next to the stair
Offset to Tread Edge	Indicates the distance between the edge of the stair (the solid structure or the stringers) and the guardrail
Top Rail	
Create Top Rail	Select whether to create a top rail on the guardrail
Profile Type	Select the basic geometry of the top rail • **Line:** A 3D polyline serves as the railing profile • **Flat:** A 3D polygon with width, but no height, serves as the railing profile • **Round:** The cross-section is round, controlled by width • **Rectangular:** The cross-section is rectangular, controlled by width and height
Width	Sets the width for flat, rough, and square top railing geometry
Height	Sets the height of rectangular top railing geometry
Start at Bottom	Defines where the top rail of the guardrail begins • **At Post:** The top rail begins at the first post • **At Tread:** The top rail begins at the stair tread • **Extend Tread by:** Sets a specific distance the top rail extends past the leading edge of the first tread or start of the stair. Select whether the bottom extension is horizontal to the bottom riser or slopes with the stair.
End at Top	Defines where the top rail of the guardrail ends • **At Post:** The top rail ends at the last post • **At Tread:** The top rail ends at the stair tread • **Extend Tread by:** Sets a specific distance the top rail extends past the leading edge of the last/top tread/end of the stair. Select whether the bottom extension is horizontal to the bottom riser or slopes with the stair.
Frame	
Create Frame	Select whether to create a frame on the guardrail
Frame Type	Sets the geometry for the area below the top rail, and between posts • **Frame:** Adds a frame to support the frame bars; the frame must include at least one of the top, bottom, or left-right vertical supports • **Panel:** Adds a solid panel below the top rail
Top/Bottom/ Left-Right Verticals	Select options to include in the frame

Parameter	Description
Offset to Top Rail	Indicates the perpendicular distance between the bottom of the top railing, and the top of the frame or panel
Offset to Stair	Sets the perpendicular distance between the leading edge of the tread/nosing or the bottom of the frame or panel
Profile Type	For Frame types, sets the frame cross-section geometry • **Line:** A 3D polyline serves as the railing profile • **Flat:** A 3D polygon with width, but no height, serves as the railing profile • **Round:** The cross-section is round, controlled by width • **Rectangular:** The cross-section is rectangular, controlled by width and height
Width	Sets the width of the frame or panel
Height	Sets the height of rectangular profiles
Frame Bars	
Create Horizontal Bars	Select whether to create horizontal frame bars
Create Vertical Bars	Select whether to create vertical frame bars
Profile Type	Sets the cross-section geometry of the frame bars • **Line:** A 3D polyline serves as the frame bar profile • **Flat:** A 3D polygon with width, but no height, serves as the frame bar profile • **Round:** The cross-section is round, controlled by thickness • **Rectangular:** The cross-section is rectangular, controlled by thickness and depth
Thickness	Sets the thickness of round and square frame bar profiles
Depth	Sets the depth of rectangular frame bar profiles
Maximum Distance	Specifies the maximum distance between centerlines of frame bar geometry, setting the spacing of the frame bars
Posts	
Create Posts	Select whether to create vertical supports of the top rail and frames
Shape	Sets the geometry of the guardrail posts • **Line:** A 3D polyline serves as the post profile • **Flat:** A 3D polygon with width, but no depth, serves as the post profile • **Round:** The cross-section is round, controlled by width • **Rectangular:** The cross-section is rectangular, controlled by width and depth
Width	Sets the width for flat, round, and rectangular posts
Depth	Sets the depth of rectangular posts
Arrangement	Determines how to place the posts
On each Tread	Places one post at the midpoint of the tread depth for each tread
Distance (approx.)	Specifies the distance between each post

Parameter	Description
Placement	Links the post positions on the left and right sides of the stair based on one of these methods: • **Each side individually:** The post positions are calculated separately for each side of the stair based on the approximate **Distance** value • **Left side has priority:** The left side post distances are calculated based on the approximate **Distance** value; the posts on the right are placed symmetrically • **Right side has priority:** The right post distances are calculated based on the approximate **Distance** value; the posts on the left side are placed symmetrically
On each Corner	Places a post at each landing corner, regardless of the distance set
Min Distance	Specifies the minimum distance between a corner post and the next adjacent post on a landing
Previews	Three preview panes display: • The detailed plan and elevation view of a representative portion of the railing system • An overall top/plan view of the stair • An isometric view of the stair, when **Component** is set to 2D and 3D on the General tab

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Stair Settings: General Tab
Stair Settings: Geometry Tab
Stair Settings: 2D Graphics Tab
Stair Settings: Construction Tab
Stair Settings: Graphic Attributes Tab
Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Stair Properties
Transferring Stair Properties
Creating a Stair
Inserting Stairs

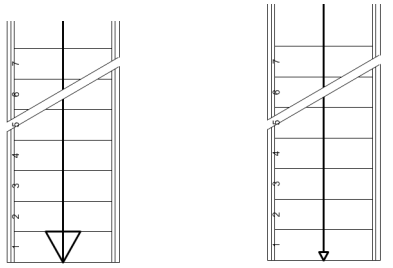
A L Stair Settings: Graphic Attributes Tab

The settings on the tab can be saved as a set and then loaded, replacing the current tab parameters.

1. Click the Graphic Attributes to specify the appearance of stair components.

Click to show/hide the parameters.

Parameter	Description
Attributes Styles	Saves and recalls styles from a saved set
View	Determines the view to apply the parameters to the stair; depending on drawing settings like layer scale and the document preference **Hide details**, the detailed or schematic view status is displayed
Detailed	Applies the graphic attributes to stairs in a detailed layer scale view
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Schematic	Applies the graphic attributes to stairs in a schematic scaled view
Accept for all Views	Applies the graphic attributes to stairs in all views (detailed and schematic)
Graphic Attribute Settings	Lists all geometry and text parameters that have graphic attributes settings, divided by 2D and 3D attributes. Click the disclosure arrow to reveal further parameters under certain categories. Each category's current class, fill, pen, line style, line thickness, opacity, and if applicable, texture is displayed. Double-click on a line to set attributes. Select a class for each stair component to control its appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the currently active class, select the default class name, or select the class named <Stair Class> which places the stair component in the same class as the stair object.
2D Walk Line and Marker Attributes	In addition to the class, pen, fill, and other "typical" attributes, the walk line and markers have specific parameters The Double Line, Oval, and Filled Arrowhead marker styles are available especially for stair walk lines to meet DIN requirements
Start Marker	Sets the graphics for the beginning of the walk line arrow that indicates the Up direction of travel on the stair, from the bottom tread, or start, of the stair
End Marker	Sets the graphics for the end of the walk line arrow that indicates the Up direction of travel on the stair, at the top, or end, of the stair
Fit to tread depth	Fits the line end marker of the walk line to the depth of the tread  Fit to tread depth enabled Fit to tread depth disabled
Set Class Style	Sets each attribute (fill, pen, line style, line thickness, opacity, and texture) of all stair components to the "By Class" setting. The attributes are then controlled by the class assigned to the component. (See "Setting Class Attributes" on page 181.) To set only the opacity by class, select the **Use Class Opacity** option when defining the graphic attributes for each category.
Remove Class Style	Removes the "By Class" setting for each attribute of all stair components. The attributes are then controlled by the individual attribute settings made for each component.
Apply to all Stairs	Applies all the attribute settings of the current stair to all stairs in the file, including symbol definitions and stairs that may be within groups. Use caution when selecting this option, as it affects all stairs in the file.

2. The graphic attributes of each item is displayed, and can be set by double-clicking on the item's line.

An attributes dialog box with the name of the item is displayed. See “The Attributes Palette” on page 1093 for information on setting attributes.

3. Set the attributes and click **OK** to return to the Graphic Attributes tab.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Stair Settings: General Tab
Stair Settings: Geometry Tab
Stair Settings: 2D Graphics Tab
Stair Settings: Construction Tab
Stair Settings: Railings Tab
Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Stair Properties
Transferring Stair Properties
Creating a Stair
Inserting Stairs

AL Managing Stair Styles

In the Stair Settings dialog box, the 2D Graphics, Construction, Railings, and Graphic Attributes tabs have the option of saving the current tab’s settings as a style that can be recalled later. The stair minimum/maximum settings can also be saved as a style set.

Saving Parameter Styles

To save tab settings as a style:

1. When the tab parameters have been set, click **Save** (located in the Styles area at the top left of the tab).

   The Save Style dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Create New Style	Saves the settings in a new style set; enter the set name
Replace Style	Replaces the selected style with the current set of parameters

2. Click **OK** to save the style set.

   The new style name appears in the Styles list.

Recalling a Saved Style

To recall a saved style:

Select the style from the Styles list. Any currently set parameters are replaced by those of the saved style set. The Styles list contains saved styles as well as a limited number of default styles.

Managing Styles

Styles can be renamed and deleted.

Default styles (located in the application folder) cannot be renamed or deleted.

To manage styles:

1. Click **Manage Styles** (located in the Styles area at the top left of the tab).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Styles dialog box opens, listing all currently saved style sets and their saved location.

Click to show/hide the parameters.

Parameter	Description
Rename	Opens the Rename Style dialog box, to rename the currently selected style name. Enter the new name for the style. Click **OK** to return to the Styles dialog box.
Delete	Deletes the selected style

2. Click **OK** to return to the Stair Settings dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Transferring Stair Properties
Inserting Stairs

AL Saving a Stair as a Symbol

The stair configuration, along with all its parameters, can be saved as a symbol for use again in the current file or in another “library” file. The stair is saved as a red symbol and becomes a plug-in object with pre-set parameters when inserted. See “Symbols” on page 237.

To save the stair as a symbol:

1. From the Stair Settings dialog box, click **Save as symbol**.

   The Save Stair dialog box opens.

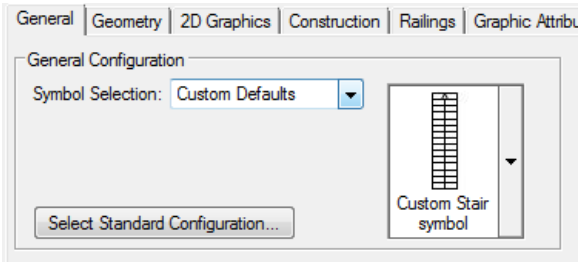
Click to show/hide the parameters.

Parameter	Description
Destination	Select whether to save the stair symbol in the current file or in a library file available in the user folder
Save as symbol in current document	Saves the stair as a symbol in the current file
Save as template in library file	Saves the stair symbol to a custom default library file located in the user folder; the exact location is displayed upon saving
Name	Enter a name for the symbol
Symbol Folder	Select the symbol folder from the available hierarchy in the current file; subfolder lists are displayed for the selected folder
Symbol folder list	Displays available subfolders for the selected symbol folder; select a location for saving the new stair symbol
New Folder	Creates a new symbol folder in the current file; provide a folder name
Browse	Browses the subfolder contents of the currently selected symbol folder

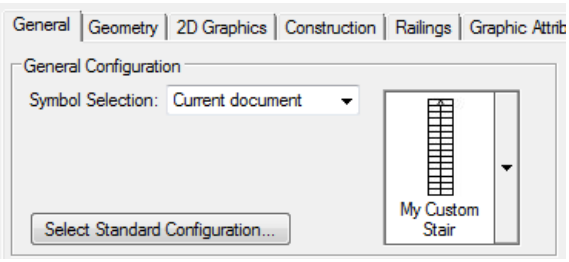
2. Select whether to save the stair symbol in the current file or in a library file.
   - To save the symbol in a library file, select **Save as template in library file** and enter a name for the symbol
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

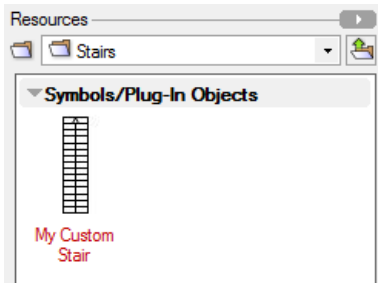
- To save the symbol in the current file, select **Save as symbol in current document**, enter a name for the symbol, and select a symbol folder destination for the symbol by double-clicking on the selected symbol folder name

3. Click **OK**.
   - If the symbol was saved in the user default library, it is available from the General tab's **Symbol Selection** list under the Custom Defaults category. Library files can be shared among users.



   - If the symbol was saved in the current file, it is available from the General tab's **Symbol Selection** list under the Current Document category, and also from the Resource Browser. Symbols can be shared by exporting them from the Resource Browser (see "Exporting Custom Resources" on page 234).





~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Managing Stair Styles
Setting Minimum and Maximum Values for Stair Geometry Parameters
Transferring Stair Properties
Inserting Stairs

AL Setting Minimum and Maximum Values for Stair Geometry Parameters

When setting stair parameters, a geometry "envelope" can be set to specify the allowable value ranges for tread depth, riser height, step length, and stair angle. Setting these values can assist with creating proportional stair geometry that complies with standards.

> During the initial phases of design, it may be convenient to temporarily disable the minimum/maximum restrictions by deselecting **Use Minimum/Maximum Values** from the Stair Settings dialog box.

The stair minimum/maximum settings can be saved as a style set and then loaded, replacing the current dialog box parameters.

To set minimum and maximum stair parameter ranges:

1. From the Geometry pane of the Stair Settings dialog box, click **Edit**.

   The Min/Max Settings dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Min/Max Styles	Saves and recalls styles from a saved set
Values	Specify the minimum and maximum values for the tread depth, riser height, and step length. Additionally, set the maximum angle (steepness) for the entire stair. For fixed angle winder stairs, set the minimum tread depth value.

2. Set the geometry minimum and maximum values and click **OK**.

Managing Stair Styles
Saving a Stair as a Symbol
Transferring Stair Properties
Inserting Stairs

Stair Properties

Certain stair parameters can be edited directly from the Object Info palette, depending on the settings of the selected stair. These parameters display when the **Total Rise** of the stair is set by Layer Elevation, or the stair has an upper floor representation. All of the stair parameters can be edited by clicking **Settings** from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Top Bound	Sets the vertical reference that determines the top of the stair. The **Layer Wall Height** value is set by the design layer (see "Setting Design Layer Properties" on page 165). Alternatively, the top of the stair can be bound by one of the story levels defined for the story or the story above it. By setting the top of the stair to a level type, if the elevation of the associated story changes, the height of the stair changes automatically to match.
Top Offset	Sets the offset of the top of the stair from its specified top bound height
Bottom Bound	Sets the vertical reference that determines the bottom of the stair. Alternatively, the bottom of the stair can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the stair to a level type, if the elevation of the associated story changes, the height of the stair changes automatically to match.
Bottom Offset	For the bottom of the stair, sets the offset from its specified bottom bound height
Upper Layer	If an upper floor representation of the stair is enabled, specify the layer to display the representation.
Settings	Opens the Stair Settings dialog box, to edit all of the stair parameters

Stair Settings: Geometry Tab
Stair Settings: 2D Graphics Tab
Transferring Stair Properties
Setting Design Layer Properties
Creating a Stair

Transferring Stair Properties

Specific stair parameters can be transferred from one stair to another.

To transfer stair attributes:

1. Click the **Stair** tool from the Building Shell tool set and select **Pick up** mode from the Tool bar.
2. Click **Preferences** from the Tool bar to set the transfer parameters.

   The transfer attributes cannot be accessed when clicking **Settings** from the Object Info palette.
3. Click the Transfer tab.

Click to show/hide the parameters.

Parameter	Description
Attributes	Select the stair attributes to transfer; selected attributes display with a check mark
All/None	Selects/deselects all the attributes with a single click
Picked up values become stair defaults	When the selected attributes are picked up in **Pick up** mode, they also become the default parameters for the **Stair** tool
Accept for all Stairs in Drawing	The most recently picked up, selected attributes, are applied to all stairs in the file

4. Select the attributes to transfer, and select whether the attributes should become **Stair** tool defaults and/or should also apply to all stairs in the file.
5. Click **OK**.
6. In **Pick up** mode, click on the stair with the desired attributes.

   The cursor changes to an eyedropper, and the source stair is highlighted in red.
7. Move the cursor to the target stair, and click **Apply** mode from the Tool bar.

   Press the Option (Mac) or Ctrl (Windows) key to switch between the Pick up and Apply modes.
8. Click on the target stair to transfer the selected attributes.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Managing Stair Styles
Saving a Stair as a Symbol
Setting Minimum and Maximum Values for Stair Geometry Parameters
Inserting Stairs

Inserting Escalators

A basic escalator can be inserted, with its height defined automatically from its top and bottom layer boundaries.

To insert an escalator:

1. Select the **Escalator** tool from the Building Shell tool set.
2. The escalator is a point object; click once to insert the escalator, and then again to set the rotation.

   If this is the first time the tool is used in this session, the Escalator Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The parameters can be changed in the Object Info palette after insertion.

Click to show/hide the parameters.

Parameter	Description
Height	
Height	Directly sets the desired height of the escalator. When the escalator height is determined manually by this method, the **Top Bound** property of the escalator is automatically set to Layer Elevation, and the **Top Offset** value is modified accordingly. When the top of the escalator is bound by the layer wall height value or by a story level, the escalator height displays automatically.
Top Bound	Sets the vertical reference that determines the top of the escalator. The **Layer Wall Height** value is set by the design layer (see "Setting Design Layer Properties" on page 165). Alternatively, the top of the escalator can be bound by one of the story levels defined for the story or the story above it. By setting the top of the escalator to a level type, if the elevation of the associated story changes, the height of the escalator changes automatically to match.
Top Offset	Sets the offset of the top of the escalator from its specified top bound height
Bot Bound	Sets the vertical reference that determines the bottom of the escalator. Alternatively, the bottom of the escalator can be bound by one of the story levels defined for the story or the story below it. By setting the bottom of the escalator to a level type, if the elevation of the associated story changes, the height of the escalator changes automatically to match.
Bot Offset	For the bottom of the escalator, sets the offset from its specified bottom bound height
Type	Select a generic escalator or one of the common escalator types
Slope angle	Select from the common slope angles available for an escalator
Floor depth above	Indicates the depth of the floor-to-ceiling construction on the upper floor being serviced by the escalator; this parameter affects the size of the headroom and pit in Top/Plan view
Step width	Select a common step width configuration
Horizontal lead in	Select the number of horizontal escalator steps at the bottom of the escalator
Balustrade type	Select the type of geometry that encloses the sides of the escalator and supports the moving handrail
Section cut	Select whether to display the escalator with a section break going up or down, or without a break
Draw 3D	In 3D views, represents the escalator with a 3D polygon
Show pit	Indicates whether the extents of the minimum required pit below the escalator display in Top/Plan view
Show headroom	Indicates whether the extents of the minimum required headroom above the escalator display in Top/Plan view

Parameter	Description
Override Maximum Allowable Span	Overrides the maximum allowable length of the selected escalator type; this parameter is affected by the horizontal lead-in and slope angle values, as well as the selected escalator type
Override Maximum Height Limit	Overrides the maximum allowable floor-to-floor height of the selected escalator type; this parameter is affected by the horizontal lead-in and slope angle values, as well as the selected escalator type

Inserting Ramps

To insert a ramp:

1. Select the **Ramp** tool from the Building Shell tool set.
2. The ramp is a point object; click once to place the lower end of the ramp, and then again to set the rotation.

   If this is the first time the tool is used in this session, the Object Properties dialog box opens. Specify the preferences to use for this tool during this session, or accept the defaults.

   When the ramp configuration has been selected, the available parameters in the dialog box change to reflect the selection.
3. Click **OK**.

   The parameters can be changed in the Object Info palette after insertion.

Click to show/hide the parameters.

Parameter	Description
Style	Select a either a standard (simple board) or masonry ramp style
Width	Set the ramp's width
Height	Set the ramp's height
Rise over Run	Set the ramp's rise over run value. The rise is the distance along the Y axis the ramp rises above the Z height, and run is the distance along the X axis for the ramp to reach that height. This value can be entered as a fraction, and the Vectorworks software will convert it to a decimal.
Thickness	For a standard style ramp, set the ramp's thickness
Config	Select the ramp's configuration from the list of preformatted options
Landing Height	Set the landing's height above the finished floor, if applicable
Offset 1	For ramps with a landing, set a distance, if applicable, to offset the bottom of the upper ramp away from the top of the lower ramp
Offset 2	For ramps with a landing, set a distance, if applicable, to offset the bottom of the upper ramp toward the top of the ramp
Curved landing	Select to make the landing curved
2D Stair Break	Select to create a break line in the 2D view
Create 3D	Select to create a 3D view

Parameter	Description
Left Rail	Select to provide a handrail on the left side (facing the top of the ramp)
Right Rail	Select to provide a handrail on the right side (facing the top of the ramp)
Rail H	Set the handrail height above the ramp surface
Rail W	Set the handrail width
Arrows (2D view)	For 2D views, select whether to provide an arrow pointing up or down, or no arrow

Inserting Handrails

Handrails can be inserted independently of those available as part of the **Stair** tool.

 To insert a straight handrail:

 To insert a curved handrail:

1. Select either the **Straight Handrail** tool or **Curved Handrail** tool from the Furn/Fixtures tool set.
2. The handrails are point objects; click once to place the handrail and then again to set the rotation.

   If this is the first time the tool is used in this session, the Object Properties dialog box opens. Specify the preferences to use for this tool during this session, or accept the defaults.
3. Click **OK**.

   The parameters can be changed in the Object Info palette after insertion.

Click to show/hide the parameters.

Parameter	Description
Length	For a straight handrail, set the overall length
Overall Angle	For a curved handrail, set the overall angle of the curve
Radius	For a curved handrail, set the radius of the curve
Top Rail Width	Set the width of the top rail
Top Rail Thk	Set the thickness (height) of the top rail
Inter Rail Width	Set the width of the lower horizontal rails
Inter Rail Thk	Set the thickness (height) of the lower horizontal rails
Starting Upright	Select to place an upright flush with the start of the horizontal rails
Ending Upright	Select to place an upright flush with the end of the horizontal rails
1st Upright Spacing	Set the distance before the first upright (not counting the starting upright, if one is included)
Upright Spacing	Set the spacing, measured on center, between all subsequent uprights
Upright Width	Set the width of the uprights
Upright Depth	Set the depth of the uprights
Caps	Select to add caps to one or both ends

Parameter	Description
1st Horizontal Height/ Spacing	Set the height/spacing of the first horizontal rail (not counting the top rail), measured down from the top of the handrail
Horizontal Spacing	Set the spacing, measured on center, between all subsequent horizontal rails
Overall Height	Set the overall height of the handrail
Rise	Set the rise from one end of the handrail to the other
Start Miter	For a straight handrail, set the angle in degrees of the miter at the beginning of the top rail
End Miter	For a straight handrail, set the angle in degrees of the miter at the end of the top rail
Top Rail/Uprights/ Intermediate Rails	Select the class from the current list of classes, or select **New** to create a new class (see “Creating Classes” on page 177)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Stair Settings: Railings Tab
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Furniture and Fixtures

Inserting Cabinets

All Vectorworks products include three types of cabinets for inserting in a drawing file—base, wall, and utility cabinets. In the Vectorworks Fundamentals product, the cabinet objects are inserted from an object library through the Resource Browser. In the Vectorworks Design Series products, the cabinet objects are inserted with the various cabinet tools. Cabinet door and drawer handles are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219).

 To insert a base cabinet:

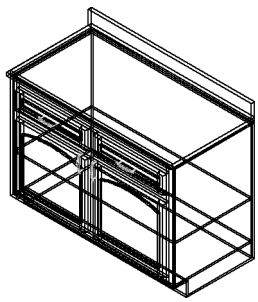
 To insert a utility cabinet:

 To insert a wall cabinet:

1. Select the **Base Cabinet** or **Utility Cabinet** or **Wall Cabinet** tool from the Furn/Fixtures tool set.
2. Click in the drawing file or wall to set the insertion point of the cabinet, and click again to set the rotation.

   If this is the first time a cabinet of this type is placed in the drawing, the Object Properties dialog box opens.
3. Specify the default properties which apply to cabinets subsequently placed in this document. Cabinet properties can be edited later in the Object Info palette.

Base Cabinet Parameters

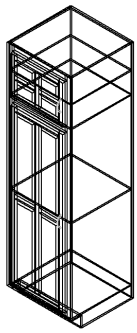


Click to show/hide the parameters.

Parameter	Description
Flip	When inserting a cabinet in a wall, click to flip the cabinet orientation
Position	When inserting a cabinet in a wall, click to set the position of the cabinet; in the Position Symbol in Wall dialog box, enter the distance from the symbol’s insertion point to the selected corner of the wall
Style	Select the cabinet style
End Finish	Select the cabinet finish
Blind	Specify the direction of the blind, or none
Length	Enter the cabinet length
Uneven Length Corner	Select whether the cabinet has an uneven length corner, and if so, enter the left corner length
Height / Depth	Enter the cabinet height and depth

Parameter	Description
Door mounting	Select the cabinet door mounting style
Draw Drawer	Select whether to draw the cabinet drawer(s)
Drawer Style	Specify the drawer style
Number of Doors	Select the number of cabinet doors
Door Style	Select the cabinet door style
Panel Style	Select the door panel style
Arch Height Factor	Specify the proportion of the arch height in relation to the door width (a larger value produces a taller arch)
Door Swing	Specify the door swing direction
Rail and Stile Width	Select the rail and stile width
Bevel Edge	Select whether to bevel the outside door and drawer edges
Bevel Inside	Select whether to bevel the inside door and drawer edges
Reveals	Enter the Top, Mid, Bottom, Side, and Center reveal height or width
Number of Shelves	Enter the number of shelves
Draw Kick	Select whether to draw a kick
Kick Height / Depth	Specify the kick height and depth
Draw Counter	Select whether to draw a counter
Back Splash	Specify where to place the back splash, or none
Splash Height	Enter the back splash height
Splash Thickness	Enter the back splash thickness
Counter Thickness	Enter the counter thickness
Overhang	Enter the counter overhang length
Reveal	Enter the thickness for the bottom portion of the counter
Show Detail	Select whether to draw various cabinet details. Select the cabinet door, kick, and/or counter class, or none
Door Handle Height	Enter the door handle height
Door / Drawer Handles	Click **Door Handles** or **Drawer Handles** to select a handle from the default content (see “Resource Libraries” on page 219); select a symbol from the graphical list of **Symbols** and click **OK** (or select the No Pull symbol to remove the door or drawer handles)

Utility Cabinet Parameters

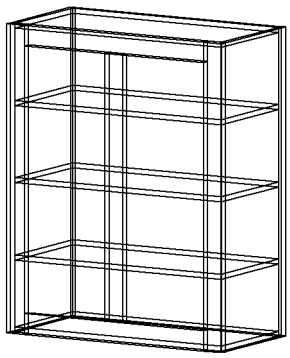


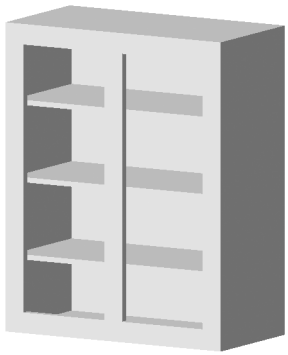
Click to show/hide the parameters.

Parameter	Description
Flip	When inserting a cabinet in a wall, click to flip the cabinet orientation
Position	When inserting a cabinet in a wall, click to set the position of the cabinet; in the Position Symbol in Wall dialog box, enter the distance from the symbol's insertion point to the selected corner of the wall
Length	Enter the cabinet length
Height / Depth	Enter the cabinet height and depth
Door Mounting	Select the cabinet door mounting style
Number of doors	Select the number of cabinet doors
Door Swing	Specify the door swing direction
Door Style	Select the cabinet door style
Panel Style	Select the door panel style
Mullion Style	Select the cabinet mullion style, or none
Handle Height	Enter the door handle height
Number of Shelves	Enter the number of shelves
Upper Door Height	Enter the upper cabinet door height
Upper Door Style	Select the upper cabinet door style
Upper Panel Style	Select the upper cabinet door panel style
Upper Mullion Style	Select the upper cabinet mullion style, or none
Upper Handle Height	Enter the upper cabinet door handle height
Upper Number of Shelves	Enter the number of shelves in the upper cabinet
Arch Height Factor	Specify the proportion of the arch height in relation to the door width (a larger value produces a taller arch)
Rail and Stile Width	Select the rail and stile width
Bevel Edge	Select whether to bevel the outside door and drawer edges
Bevel Inside	Select whether to bevel the inside door and drawer edges
Reveals	Enter the Top, Mid, Bottom, and Side reveal measurement height or width

Parameter	Description
Draw Kick	Select whether to draw a kick
Kick Height / Depth	Specify the kick height and depth
Show Detail	Select whether to draw various cabinet details. Select the cabinet, door, kick, and/or glazing class, or none
Handles	Click **Handles** to select a handle from the default content (see "Resource Libraries" on page 219); select a symbol from the graphical list of **Symbols** and click **OK** (or select the No Pull symbol to remove the handles)

Wall Cabinet Parameters





Click to show/hide the parameters.

Parameter	Description
Flip	When inserting a cabinet in a wall, click to flip the cabinet orientation
Position	When inserting a cabinet in a wall, click to set the position of the cabinet; in the Position Symbol in Wall dialog box, enter the distance from the symbol's insertion point to the selected corner of the wall
Style	Select the cabinet style
End Finish	Select the cabinet finish
Blind	Specify the direction of the blind, or none
Length	Enter the cabinet length
Uneven Length Corner	Select whether the cabinet has an uneven length corner, and if so, enter the **Left Corner Length**
Height / Depth	Enter the cabinet height and depth
AFF	Enter the cabinet height above the finished floor
Door mounting	Select the cabinet door mounting style
Number of Doors	Select the number of cabinet doors
Door Swing	Specify the door swing direction
Door Style	Select the cabinet door style
Panel Style	Select the door panel style

Parameter	Description
Mullion Style	Select the cabinet mullion style, or none
Number of Shelves	Enter the number of shelves
Arch Height Factor	Specify the proportion of the arch height in relation to the door width (a larger value produces a taller arch)
Handle Height	Enter the door handle height
Rail and Stile Width	Select the rail and stile width
Bevel Edge	Select whether to bevel the outside door and drawer edges
Bevel Inside	Select whether to bevel the inside door and drawer edges
Reveals	Enter the Top, Bottom, Side, and Center reveal measurement height or width
Glazing	Select the glazing class or none
Show Detail	Select whether to draw various cabinet details. Select the cabinet and/or door class or none
Handles	Click **Handles** to select a handle from the default content (see “Resource Libraries” on page 219); select a symbol from the graphical list of **Symbols** and click **OK** (or select the No Pull symbol to remove the handles)

4. Click **OK** to set the cabinet parameters and close the Object Properties dialog box.

Inserting Toilet Stalls

The **Toilet Stall** tool provides you with a flexible tool for creating a variety of toilet stalls for public restrooms. Toilet stall fixtures are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219).

To insert a toilet stall:

1. Select the **Toilet Stall** tool from the Furn/Fixtures tool set.
2. Click in the drawing file or wall to set the insertion point of the toilet stall, and click again to set the rotation. The direction of the toilet stall can be changed later by clicking **Flip** in the Object Info palette or from the context menu if the toilet stall is inserted in a wall, or by using the context menu commands to rotate the toilet stall if it is not.

   If this is the first time a toilet stall is placed in the drawing, the Object Properties dialog box opens. Specify the default properties which apply to toilet stalls subsequently placed in this document. Toilet stall properties can be edited later in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Number of Stalls	Select the number of stalls to place together as a single, grouped object
Stall Width	Enter the stall width
Stall Depth	Enter the stall depth
Partition Thickness	Enter the thickness for the stall partitions

Parameter	Description
Clg Height (Floor and Clg Mtd or Ceiling Mtd only)	Enter the height for ceiling-mounted stall supports
Partition Type	Select the mounting type for the stall partitions If none is selected for **Partition Type**, all parameters related to partitions, doors, or grab bars will not have any visible effect on the toilet stall
Left Partition	Deselect to remove the left partition from the stall. If you have multiple stalls, this will only remove the farthest left partition.
Dividing Partition	Select to include partitions between each stall
Right Partition	Deselect to remove the right partition from the stall. If you have multiple stalls, this will only remove the farthest right partition.
Front Partition	Deselect to remove the front partition from the stall. If the door is set to the front partition, the door is also removed.
Door	Select to include doors on each stall
Door Closed in 3D	Select to display the stall door as closed in 3D views. Deselect to have the stall door open at an angle.
Show Door In	Select which partition for the door to be a part of, if any. Doors cannot be set to the right or left partitions if there are multiple stalls in the toilet stall group.
Door Swing	Select the direction the door opens
Door Width	Enter the door width
Door Offset	Enter the door offset from the partition
Fixture Position	Select the position of the fixture within the stall
Show 2D Accessible Icon	Display the accessible symbol in 2D view
Show Turning Circle	Display the turning circle for wheelchair access in 2D view The turning circle requires a minimum clear space in the stall 62” wide by 60” deep
Left/Rear/Right Grab Bar	Select to place a grab bar on the stated side of the stall
Show Fixture	Select to show the toilet fixture
Choose Fixture	Select the desired toilet fixture from either the default content or the current file’s content. Select a symbol from the graphical list of **Symbols** and click **OK**.

3. Click **OK** to set the toilet stall parameters and close the Object Properties dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbols in Walls
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

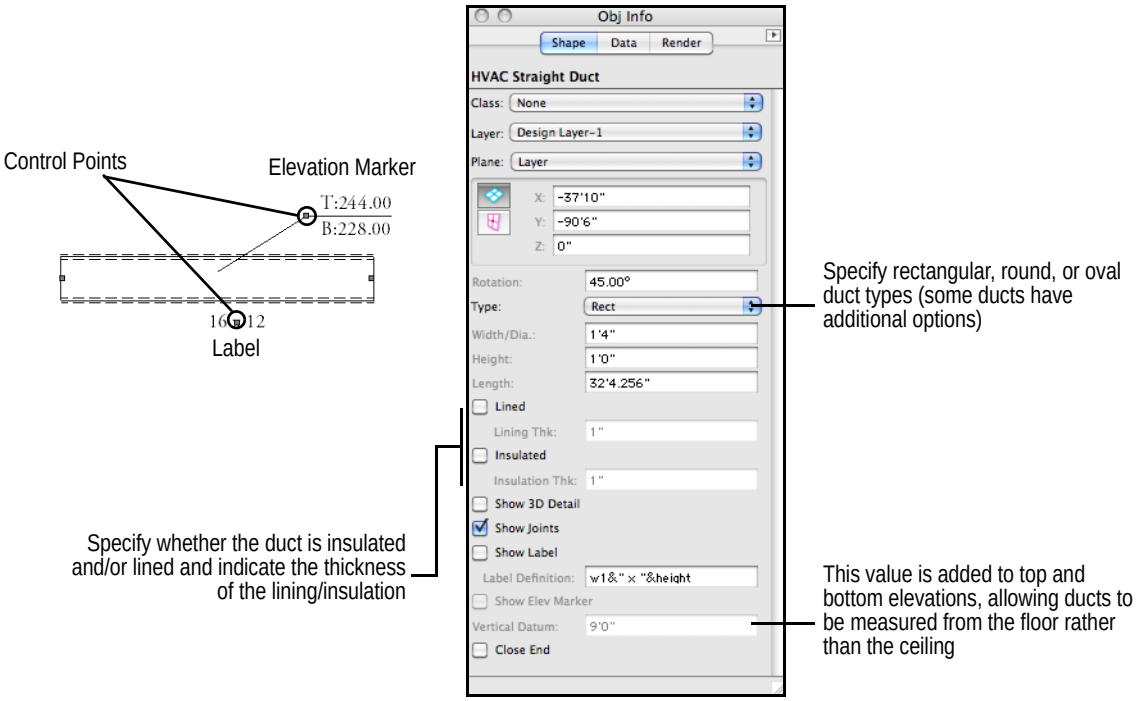
MEP Objects

HVAC

The Heating, Ventilating, and Air Conditioning (HVAC) tools and objects available in the Vectorworks Architect product add HVAC ducts and diffusers in a variety of configurations to a drawing. These geometric models can represent a variety of actual items. For example, the diffuser object can have its dimensions and parameters set to develop many differently sized diffusers, including industry standard sizes that can be predefined.

HVAC objects are available in [Vectorworks]\Libraries. These objects are accessed through the Resource Browser and consist of hybrid 2D/3D representations of common HVAC duct work elements. See “Using the Resource Browser” on page 221.

Duct work objects include optionally displayed labels and elevation marker tags. The label contents are determined by a user-defined string of values, such as width and height (obtained automatically) and any text desired. The elevation marker tag indicates the top and bottom above the finish floor of an HVAC duct work item; **Show 3D Detail** must be enabled to display the elevation marker tag. Both the HVAC label and elevation marker tag can be easily moved by dragging its control point to the correct location.



A diffuser report, containing a diffuser, register, and grille schedule, can be included. It can be added to the drawing from the **VA Create Schedule** command or the Resource Browser. From the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder that is included with the Vectorworks Architect product. Drag the Diffuser Report worksheet to the drawing. An HVAC Diffuser, Register and Grille worksheet, populated with information from the objects in the current drawing, is automatically created.

For more information on labels, elevation marker tags, and HVAC reports, refer to the Duct_Object.pdf file included in [Vectorworks]\Extras.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Resource Libraries
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Electrical and Communication Circuiting

The communications and electrical circuiting tools and objects available in the Vectorworks Architect product add electrical and communication objects, including panels and disconnects, to a drawing. The circuiting tools link these objects, creating a communication or electrical circuit.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Electrical and Communication Objects

Electrical and communication objects are available from the MEP tool set and in specific object libraries (see “Resource Libraries” on page 219). These objects are accessed through the Resource Browser, and consist of hybrid 2D/3D representations of common electrical devices. These symbols already have the correct records attached for proper circuiting. See “Using the Resource Browser” on page 221.

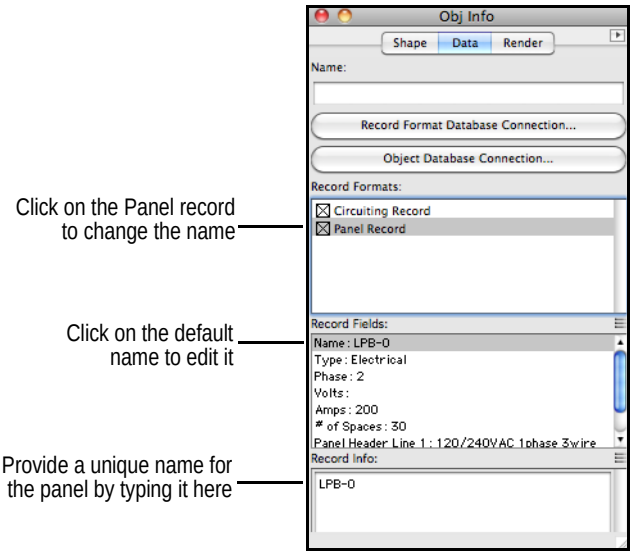
It is also possible to create custom electrical symbols, panels and disconnects; attaching the correct record to these symbols makes them circuitable. See “Creating Custom Electrical and Communication Symbols” on page 683 for information.

Naming Panels

When inserting panels (electrical or communication) from object libraries, give each panel its own name so that it can be uniquely identified by the **Circuiting** tool.

To identify an inserted panel with a unique name:

1. Insert the panel from the Resource Browser.
2. With the panel still selected, click the Data tab in the Object Info palette. Provide the unique panel name under the Panel record information.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Circuiting Tool

Once electrical or communication panels and objects have been inserted, the **Circuiting** tool links the objects by assigning each item a circuit number and associating it with a panel board.

The symbol or object that is being circuited must be on a visible (not grayed) layer. If the symbol is not on the active layer, the symbol is copied to the active layer, and then the record information is attached. This is designed so that an electrical engineer can use workgroup referencing to access architectural layers with preliminary outlet locations. The engineer can then copy those outlets to the proper layer without re-entering data.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Circuit
Adding to a Circuit

Creating a Circuit

 To create a circuit:

1. Click the **Circuiting** tool from the MEP tool set.
2. Click on the first item to be circuited.

   If selecting a custom symbol without a Circuiting record attached, the tool attaches the record to the current symbol, as well as to all subsequent symbols in the circuit. In this case, specify whether an electrical or communication circuit is being created.

   When creating a circuit, do not include the panel symbol.

   Based on the item's record, the tool determines whether this is an electrical or communication device. The Circuiting Tool dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Circuit Information	Enter the circuit information (used on the panel schedule)
Panel	Select the associated panel
Circuit #	Enter the circuit number
Trip	Enter the trip rating
Wire Size	Enter the size of the wire
Conduit Size	Enter the conduit size
V.A./Watts	Enter the total V.A./Watts rating
Remarks	Enter remarks for the circuit
Check Voltage and Phase of Circuit Items	Select to verify that the voltage and phase of circuited items is correct
Draw circuiting lines	Select to draw connecting circuiting lines
Overwrite existing object data	Select to apply the current object information to all objects in the circuit
Object Data	Enter parameters for the object being circuited
Phase/Pole	Enter the number of phases/poles
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Voltage	Enter the voltage
V.A./Watts	Enter the V.A./Watts for this object
Remarks	Enter remarks for the object

3. Enter the desired information for the circuit and the objects in the circuit, and click **OK**.

   The information is transferred to the object's record.

4. Continue clicking on the items to be circuited in the order the circuit loop should be drawn.
5. To complete the circuit, click in an empty area of the drawing. Confirm that you wish to place the Home Run marker.

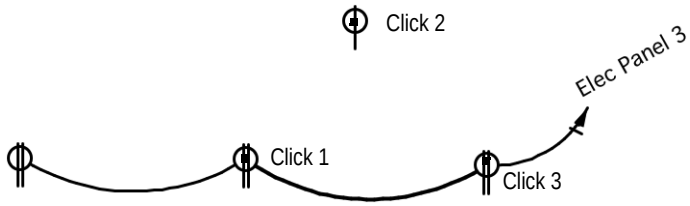
   The **Circuiting** tool generates a Home Run marker and phases are denoted as specified.

A Adding to a Circuit

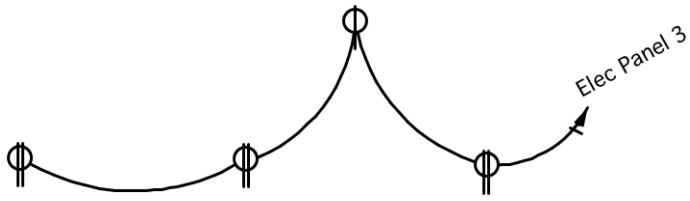
To add an item to an existing circuit:

1. Click the **Circuiting** tool from the MEP tool set.
2. Click the communication or electrical device from which to begin the new circuit connection.

   A notice displays, asking if additional items are to be added to this circuit. Click **Yes**.



3. Click the item to add to the circuit, and then select the next item in the circuit.



To complete the circuit, click away from any selectable objects. A notice displays asking if you wish to continue. Click **No** to end this procedure.

If adding an item at the end of a circuit, place the third click anywhere on the drawing. The item is chained to the item that previously had the Home Run marker attached. A notice displays, confirming whether to place the Home Run. Click **Yes**. The **Circuiting** tool generates a Home Run marker and phases are denoted.

Editing Circuits

The **Edit Circuiting** tool modifies existing circuit information. Use this tool to reassign circuit numbers and update the information attached to the circuited elements.

 To edit existing circuits or a circuited item:

1. Click the **Edit Circuiting** tool from the MEP tool set.
2. Click on the circuited item to be edited.

   The Edit Circuit Tool dialog box opens.

Click to show/hide the parameters.

Parameter	Description
List of editing operations	Selects the editing operation to perform. Select **Edit circuit information only** to edit the information which applies to the entire circuit. Select **Edit selected item only** or **Edit all items on the circuit** to enter edits which apply only to the selected circuiting item or edit both circuit and circuited object information. Select **Delete selected item only** or **Delete entire circuit** to delete the circuited item from the circuit or delete the entire circuit (the circuited items remain on the drawing).
Circuit to Edit	Selects the specific circuit to edit; available if selected item has multiple circuit assignments associated with it
Circuit Information	Enter the circuit information (used on the panel schedule)
Panel Name	Select the associated panel
Circuit #	Enter the circuit number
Trip	Enter the trip rating
Wire Size	Enter the size of the wire
Conduit Size	Enter the conduit size
V.A./Watts	Enter the total **V.A./Watts** rating
Remarks	Enter remarks about the circuit
Check Voltage and Phase of Circuit Items	Select to verify that the voltage and phase of circuit items is correct
Item Data	Enter parameters for the object being circuited
Phase/Pole	Enter the number of phases/poles
Voltage	Enter the voltage
V.A./Watts	Enter the **V.A./Watts** for this object
Remarks	Enter remarks for the object

3. Select the operation to perform.

   If there are multiple records attached to this item, select the record to edit from the **Circuit To Edit** list. The record information displays in the Circuit Tool dialog box.

4. Click **OK**.

Comm Device Tool

The **Comm Device** tool inserts an indicator and basic specifications to indicate the type and location of a communications device.

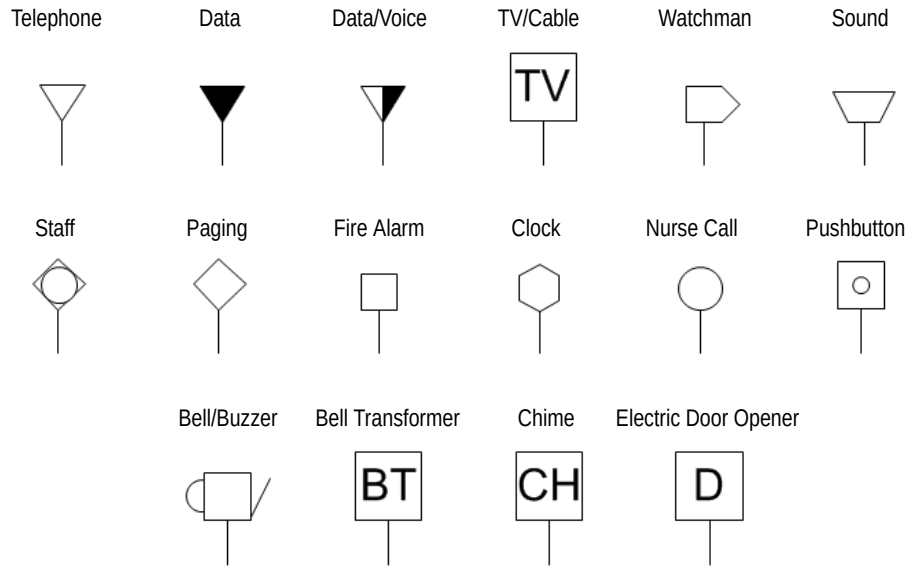
To insert a communications device:

1. Select the **Comm Device** tool from the MEP tool set.
2. Click to place the object into a wall and click again to set the rotation of the object.

   If this is the first time a communications device has been placed on the drawing, the Object Properties dialog box opens. Before placing the object, specify the preferences to use for this tool during this session, or accept the default values.
3. Click **OK**.

   Object instances can be modified in the Object Info palette after insertion.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
2D Scale Factor	Set the size of the communications device indicator; increase the scale value to obtain a larger object. Indicators maintain a constant size regardless of the drawing scale.
Config	Select the configuration of the communications device, which is indicated by a preformatted shape 
Config Label	Type the label for the communications device
Mtg Height AFF	Set the mounting height above the finished floor

Parameter	Description
Show 3D Detail	Select to show a communications device wall plate in 3D views. If **Show 3D Detail** is not selected, a 3D locus will indicate the device's location in 3D views.

Electrical and Communication Objects

Sizing Calculators

The Vectorworks Architect product provides sizing calculators for performing calculations and adding drawing specifications without leaving the program.

The software includes a standard database of wire specifications used for the Conductor Sizing and Conduit Sizing calculators. The database contains the names, dimensions, and physical properties of a selected group of wiring types.

Conductor Sizing Calculator Command
Conduit Sizing Calculator Command
Creating Piping Runs

Conductor Sizing Calculator Command

The Conductor Sizing Calculator reports the correct wire size for a given load, based on a single conductor and the length of travel.

To perform conductor sizing calculations:

1. Select **AEC > Electrical > Conductor Sizing Calc**.

   The Conductor Sizing Calculator dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Calculate	Select the type of calculation to perform
Phase Factor	Specifies the phase factor
Wire Properties	Enter the known conductor parameters; values returned by the calculation will appear dimmed
Type/Manuf.	Select the wire type or manufacturer
Size/Part Number	Select the wire size or the manufacturer part number
Sectional area of the wire, cmil	Enter the cross sectional area of the conductor wire in cmils
Approximate AWG	Enter the approximate AWG (American Wire Gauge) of the conductor
Resistance of conductor material at 75c	Enter the resistance of the conductor material at 75c
Temperature (c)	Enter the operating temperature in Celsius
Amps	Enter the amp rating
Length (ft.)	Enter the conductor length in feet

Parameter	Description
Type	Select whether the conductor type is **Aluminum** or **Copper**, and then enter an **Alpha** value if other than standard
Voltage Drop	Select whether the voltage drop is a **Percentage** or **Absolute**. If a percentage, enter the value.
Voltage	Enter the working voltage
Calculate	Performs the selected calculation

2. Select the type of calculation to perform, based on the current parameters, from the **Calculate** list.

Calculation	Description
Wire Size	Provides the sectional area of the wire (cmil) and the approximate AWG for wire sizes below 250 kcmil
Ohms/ft/cmil-ft	Calculates the resistance per sectional area of wire
Voltage Drop	Calculates the voltage drop across the length of the conductor, either as a percentage or absolute value
Phase Factor	Provides the phase factor value of the conductor
Voltage	Calculates the voltage value of the conductor
Ampacity	Determines the amp rating of the conductor
Length of Run	Calculates the allowable length of the conductor specified for the load given

3. Enter the required values in the fields. Fields where the result will be displayed appear dimmed. Some calculations require the selection of the wire type or manufacturer, as well as the wire size or part number. When all the required parameter values are complete, the **Calculate** button becomes available; click to perform the calculation.
4. The calculation results are displayed. Select another calculation to perform or click **Close** to exit.

A Conduit Sizing Calculator Command

The Conduit Sizing Calculator reports the proper size conduit(s) required for a given set of wires.

To perform conduit sizing calculations:

1. Select **AEC > Electrical > Conduit Sizing Calc**.

   The Conduit Sizing Calculator dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Conductor Item	
Type/Manufacturer	Using the information contained in the wire specifications database, select a wire type or manufacturer
Size/Part Number	Select a wire size or part number
Number of conductors	Enter the number of conductors of this type and size

Parameter	Description
C-S Area of conductor in sq inches	The cross-sectional area of the specified conductor displays
Add >>	Click to include the specified conductors in the conduit
Remove	Click to removed a selected conductor from the **Conductors Currently in Conduit** list
Clear All	Click to remove all conductors from the **Conductors Currently in Conduit** list
Conductors Currently in Conduit	Displays a list of conductors currently included in the conduit
Conduit Type	Select a conduit type
Number of conduits	Specify the number of conduits
Conduit stuff percentage	Specify a conduit stuff percentage; the percentage suggested by the software can be modified
Results	
Actual Conduit Size required in inches	Displays the actual size of conduit required
Nominal Conduit (Trade) size required in inches	Displays the nominal (trade) size of conduit required

2. When calculations are complete, click **Close**.

Creating Piping Runs

Determine the section lengths and amount of material required for piping runs. Piping runs can be drawn by clicking the **Piping Run** tool from the MEP tool set, by drawing a polygon and selecting the **Create Piping Runs** command, or by drawing a polyline and selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

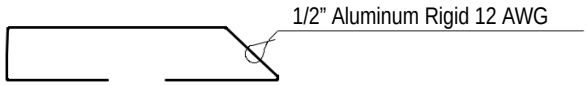
To create a piping run:

1. Draw one or more polygons to represent the piping run.
2. With the polygon(s) selected, choose **AEC > Electrical > Create Piping Runs**.

   The piping run results are displayed and can be edited in the Object Info palette. An existing piping run can be reshaped with the **Reshape** tool, and the path polyline can be edited by selecting **Edit** from the context menu (see “Object Editing Mode” on page 1004).

Click to show/hide the parameters.

Parameter	Description
Conduit Size	Specifies the diameter of the conduit
Material	Select the piping run material from the list; this selection can be displayed in the piping run tag
Remarks	Specify any remarks to be displayed in the piping run tag

Parameter	Description
Show Tag	Select to display the piping run **Conduit Size** and **Material** 
Show Remarks	Select to display the piping run **Remarks**; **Show Tag** must be selected in order to display the remarks
Length	Displays the length of the piping run
Section Length	Specifies the length of piping run sections; edit this parameter to determine various piping run requirements
Num of Runs	Specify the number of runs; increase the number to estimate the additional material required for a project
Num of Sections	Displays the number of sections required
Couplings	Displays the number of couplings required
Turns	Specifies the type of turns to use for the piping run corners
Use Bends	Bends the conduit and includes the bend in the length
Use Elbows	Counts the number of 45° and 90° elbows and displays the number
Radius	Specify the bend radius
45s / 90s	Displays the number of elbow turns (when **Use Elbows** is selected)

The piping run does not count turns at a junction box (at the location of an electrical object).

A Panel Scheduling and Diagramming

Creating a Panel Schedule

The **Create Panel Schedule** command generates panel schedules based on the circuiting information assigned to the file. If necessary, modify the dimensions and layout of the panel to conform to existing office standards.

To create a panel schedule:

1. Select **Tools > Reports > Create Panel Schedule**.

   The Create Panel Schedule dialog box opens. Specify the panel schedule criteria and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Panel list	Select a panel from the list of available panels for creating the schedule
Place in worksheet	Select to place scheduling information in a worksheet resource named with the panel name
Place in Document	Select to place a panel schedule on the drawing
Choose Schedule	Indicates the location for placing the panel schedule on the drawing
Layer	Specifies the layer for placing the schedule

Parameter	Description
Class	Specifies the class for the schedule
Panel Style	Select the panel formatting style from the list
Edit	Changes the panel schedule formatting

2. If placing the schedule on the drawing, click at the desired location.

400A 3 Phase 4 Wire
Recessed Room 102

Watt	Pole	Trip	Con	Wire	Remarks	Ckt	Ckt	Remarks	Wire	Con	Trip	Pole	Watt
						1	2						
2000	0	20	1/2"	#12	Receptacles	3	4						
2000					(Receptacles)	5	6	Receptacles	#12	1/2"	20	1	2000
						7	8						
						9	10		#12	1/2"	20	0	0
						11	12	Receptacles	#12	1/2"	20	1	2000
						13	14						
						15	16						
						17	18						
						19	20						
						21	22						
						23	24						
Phase 1 Total						0							
Phase 2 Total						2000							
Phase 3 Total						6000							

If placing the panel schedule in a worksheet, a worksheet is created with the panel name.

If circuiting changes are made, delete the outdated panel schedule and repeat the create panel schedule procedure.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Changing the Panel Schedule Format
Creating Panel Riser Diagrams

A Changing the Panel Schedule Format

To format a panel schedule:

1. Select **Tools > Reports > Create Panel Schedule**.

   The Create Panel Schedule dialog box opens.
2. Select the panel schedule and its location.
3. Click **Edit**.
4. The Schedule Formatting dialog box opens. Select the schedule to format from the **Schedule** list. A list of available fields and, if applicable, the schedule's printed name displays.

Click to show/hide the parameters.

Parameter	Description
Schedule	Select the schedule to format
Printed Name	Specifies the name that is printed on the schedule
Available Fields	Lists all possible schedule column items
Schedule Columns	Lists, in order, all included schedule columns
Move >	Moves a selected item from the Available Fields list to the Schedule Columns list
< Move	Moves a selected item from the Schedule Columns list back to the Available Fields list
Move Up / Move Down	Moves a selected item in the Schedule Columns list to adjust the list order
Printed Schedule Width	Specifies the total width of all schedule columns when printed
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Column Headers	Displays the current font, style, and size for header information in the schedule. Click **Change** to edit the font format.
Body	Displays the current font, style, and size for body text in the schedule. Click **Change** to edit the font format.

The columns shown in the **Column Order** list appear in the schedule in order.

Panel schedule formatting is stored as a worksheet named "Panel Schedule Fmt." This worksheet can be imported, using the Resource Browser, into other files where the same formatting is desired.

5. Click **OK** to return to the Create Panel Schedule dialog box. Click **OK** to place the schedule with the specified format.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

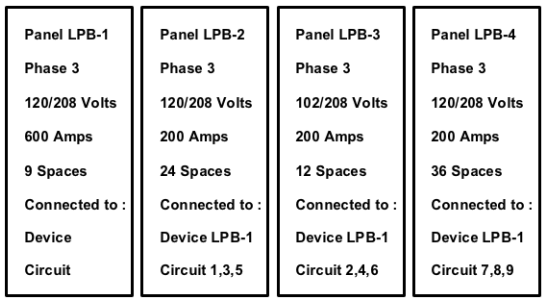
Creating a Panel Schedule
Creating Panel Riser Diagrams

A Creating Panel Riser Diagrams

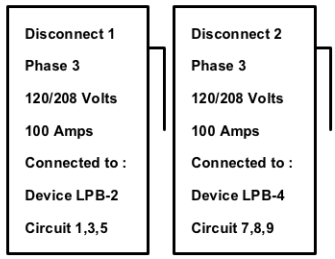
Panel riser diagrams summarize the electrical panels and disconnects in a project file.

To create a panel riser diagram:

1. Select **Tools > Reports > Create Panel Riser Diag**.
2. Click on the drawing to place the panel(s) at that location.

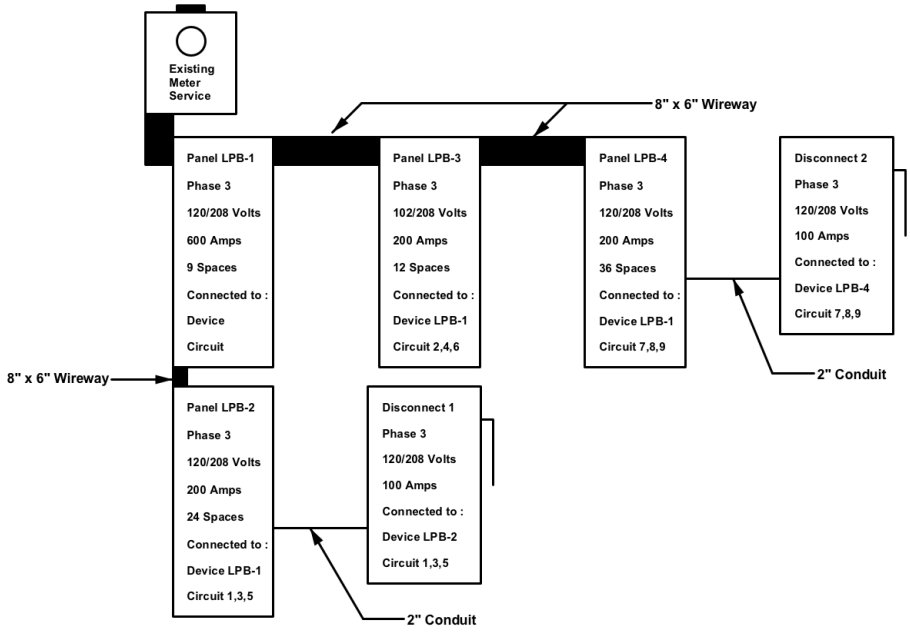


A panel displays the Panel Name, Number of Phases/Poles, Voltage, Amps, Number of spaces, Device Name, and Circuit where it is connected



A disconnect displays the Disconnect Name, Number of Phases/Poles, Voltage, Amps, Device Name, and Circuit where it is connected

Panels and disconnects can be rearranged, edited and annotated; graphics can be added to complete the panel riser diagram. If the panels are regenerated, any edits are lost. Once the panels have been generated, they can be arranged into a completed panel diagram, as seen in the following illustration:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Creating a Panel Schedule
Changing the Panel Schedule Format

Creating Custom Electrical and Communication Symbols

The Vectorworks Architect product includes many pre-defined electrical and communication objects in its object library files. However, there is no need to be restricted to these objects; creating custom panel and circuiting symbols is also possible.

Creating Custom Panels
Creating Custom Circuit Symbols

Creating Custom Panels

Electrical, communication, and disconnect panel symbols can be created specifically to fulfill a custom requirement.

To create a panel symbol:

1. Draw the object to represent the panel.
2. Convert the object into a symbol by selecting **Modify > Create Symbol**. Provide a name for the panel. See “Creating New Resources” on page 228.
3. The panel symbol requires both a panel and circuiting record. If the file does not contain panel and circuiting records, create the records according to the following formats.

Panel Record

Field	Type	Contents
Name	Text	Must be unique
Type	Text	Must be Electrical, Communication, or Disconnect
Phase	Integer	1

Field	Type	Contents
Volts	Text	117
Amps	Integer	100
# of Spaces	Integer	20
Panel Header Line 1	Text	Text placed on Panel Schedule
Panel Header Line 2	Text	Text placed on Panel Schedule

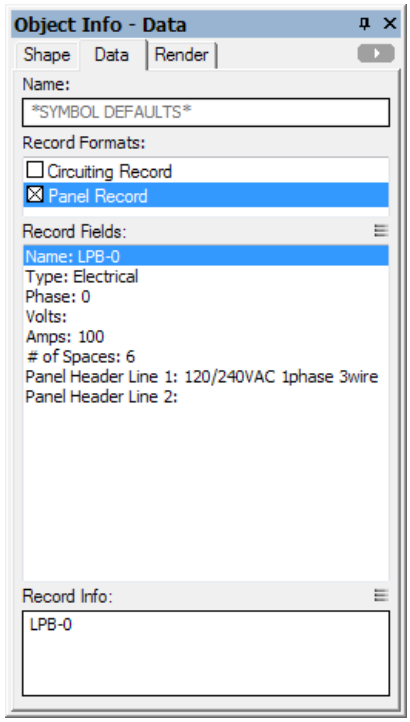
Circuiting Record

Field	Type	Contents
Name	Text	None
Circuit #	Integer	0
Wire Size	Text	0
Trip	Integer	0
Conduit Size	Text	1/2
V.A./Watts	Integer	0
Remarks	Text	Panel
Phase/Pole	Integer	2
Voltage	Text	0
Circuit Type	Integer	1 for electrical device, 2 for communication device
UID	Text	0
ID	Text	0

4. Select the symbol from the Resource Browser, and then click **Resources > Edit**.

   The Edit Symbol dialog box opens.

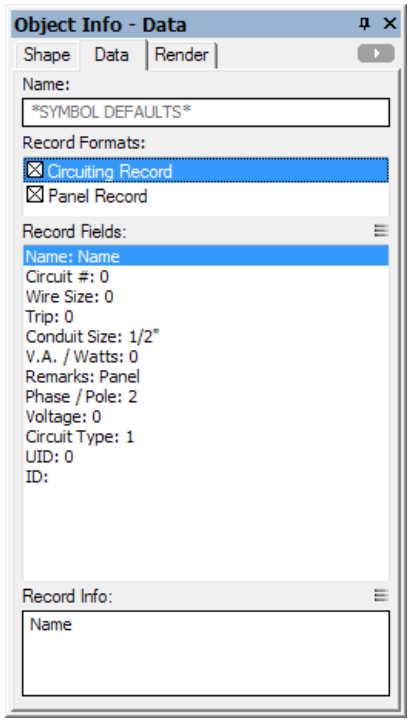
5. Click **2D Component**, and then click **Edit**.
6. In the Edit Symbol window, deselect all symbol components.
7. Attach the panel record to the symbol by selecting it from the Data tab in the Object Info palette. Enter the panel information by selecting a record field and entering its record information; the circuiting tools will use this information.



Click to show/hide the parameters.

Parameter	Description
Name	The Name field changes for each symbol later, when it is placed on the drawing
Type	Specifies Electrical, Communication, or Disconnect
Other Panel Parameters	Enter specific information about the panel
Panel Header Line 1/2	Text specified here is placed on the panel schedule

8. Attach the circuiting record to the symbol and enter the circuiting information by selecting a record field and entering its record information; the circuiting tools will use this information.



Click to show/hide the parameters.

Parameters	Description
Name	Leave this field blank
Circuit #	Enter 0 (zero) for this parameter
Other Circuit Parameters	Enter specific information about the circuit
Circuit Type	Set to 1 for an electrical symbol, and to 2 for a communication symbol
UID	Enter 0 (zero) for this parameter

9. Click **Exit Symbol** at the top right of the drawing window.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Custom Circuit Symbols

Creating Custom Circuit Symbols

Electrical and communication circuit symbols can be created specifically to fulfill a custom requirement.

To create a circuit symbol:

1. Draw the object to represent the circuitable symbol.
2. Convert the object into a symbol by selecting **Modify > Create Symbol**. Provide a name for the circuitable item. See “Creating New Resources” on page 228.
3. If the file does not contain a circuiting record, create the record according to the following format.

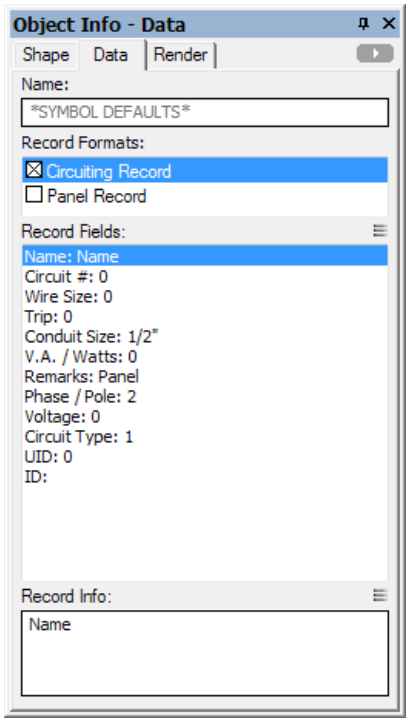
Field	Type	Contents
Name	Text	None
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Field	Type	Contents
Circuit #	Integer	0
Wire Size	Text	0
Trip	Integer	0
Conduit Size	Text	1/2
V.A./Watts	Integer	0
Remarks	Text	Panel
Phase/Pole	Integer	2
Voltage	Text	0
Circuit Type	Integer	1 for electrical device, 2 for communication device
UID	Text	0
ID	Text	0

4. Select the symbol from the Resource Browser, and then click **Resources > Edit**.

   The Edit Symbol dialog box opens.

5. Click **2D Component**, and then click **Edit**.
6. In the Edit Symbol window, deselect all symbol components.
7. Attach the circuiting record to the symbol by selecting it from the Data tab in the Object Info palette. Fill out the circuiting information by selecting a record field and entering its record information; this information is displayed and edited by the circuiting tools.



Click to show/hide the parameters.

Parameters	Description
Name	Leave this field blank
Circuit #	Enter 0 (zero) for this parameter
Other Circuit Parameters	Enter specific information about the circuit
Circuit Type	Set to 1 for an electrical symbol, and to 2 for a communication symbol
UID	Enter 0 (zero) for this parameter

8. If the symbol requires more than one circuit, create and attach a second identical record named Circuiting Record-1.
9. Click the **Exit Symbol** button located at the top right of the drawing window.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Custom Panels
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Site Modeling

A L Sitework Overview

The sitework-related commands allow the development of complex 2D and 3D models of site terrain in the Vectorworks Architect and Landmark products. The Vectorworks Landmark product contains certain additional commands.

The source data for a site model is the 3D information used to create a representation of the existing site. The information can be in the form of 3D loci, 3D polygons, or surveyor data. The site model object created from the source data contains both 2D and 3D site information, and it can be copied to other layers and files. Special "snapshot" copies of the site model allow different forms of site model to be shown side by side.

Once generated, two forms of the site model can be displayed—the existing site model and the proposed site model. The existing site model is a direct representation of the source data, including any existing site modifications. The proposed site model is the existing site model, plus the geometric effects of proposed site modifiers, such as roads and pads. The existing and proposed site models can be displayed or hidden at creation or from the Object Info palette.

Site models use the Triangulated Irregular Network (TIN) method to connect input data. This method can work with data that are scattered and in clumps—they do not have to be organized in a rectangular grid. The model uses all the data, without doing approximations, to create a network of triangles. These triangles form the terrain model; the model then performs interpolation to calculate the threading of contour lines.

After generation, the site model is cached so that updating occurs more efficiently; however, large file sizes can result. See "Setting Document Preferences" on page 60.

To upgrade a site model from a previous version of the Vectorworks program, see "Migrating from Previous Versions" on page 28.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Site Model Source Data
Creating the Site Model
Setting Site Model Properties

A L Site Model Source Data

There are four commands available to input source data into a drawing for the development of a site model; each uses a different type of source information.

- **Import Survey File**
- **2D Polys to 3D Contours**
- **3D Polys to 3D Loci** (Vectorworks Landmark only)
- **Grid Method Entry**

In addition, stake objects, 3D loci, or 3D polygons can be used directly as the source data for the site model. No conversion is necessary before using this type of source data.

The **Modify by Record** command can convert 2D polygons or polylines with attached record elevation data directly to 3D contours at the proper elevation. See "Modifying Objects by Record Value" on page 270.

Before you create the site model, check the source data with the **Validate 3D Data** command to ensure that a valid site model will be created.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing Source Data from an External File
Adding Source Data with 2D Polygons
Converting 3D Polygons to 3D Loci
Adding Source Data by Grid Method Entry
Validating 3D Source Data

Simplifying 3D Polygons

Importing Source Data from an External File

Site model source data can be generated by tabular coordinate information from an external file. This file must be a text file with fields delimited by separators in one of the formats available.

To import survey information for use as source data:

1. On the layer to receive the imported data, select **Active Layer Scale** from the drawing context menu to set the scale, or select the **File > Document Settings > Document Setup** command (see "Document Setup" on page 76).
2. Select the **Import Survey File** command from the appropriate menu:
   - Architect workspace: **AEC > Survey Input > Import Survey File**
   - Landmark workspace: **Landmark > Survey Input > Import Survey File**

   The standard Open File dialog box opens. Select the file to import.
3. The Import Survey File dialog box opens. Specify the file format options, scrolling through each line of data if necessary, and set the file import options.

Click to show/hide the parameters.

Parameter	Description
Data from selected file	Displays the first line of data in the selected file
Line #	Displays the line number of the data displayed; click the < or > button to check each line in the file
File format	Select the file format of the imported file
Units	Select the unit of the imported file
Field separator	Specifies the file field delimiter
Tab delimited	Select if fields are separated by tabs
Other character	If fields are not separated by tabs, select **Other character** and then enter the character separating each field
Import Options	
Create Stake Objects	Places stake objects to represent each 3D point in space, with text to display the elevation when in Plan view (see "Inserting Stake Objects" on page 769). Stake objects include the appropriate Easting, Northing, and Elevation values. For big sets of source data, creating large numbers of stake objects could result in performance issues. It may be preferable to create 3D loci instead.
Create 3D Locus Objects (imports only location)	Places 3D loci, which contain only location information (X, Y, and Z coordinate)

4. Click **OK**.

As each line of the file is read, either 3D loci or stake objects are inserted into the drawing at the coordinates specified by the file. If this is the first time a stake object has been inserted in the drawing, the Object Preferences dialog box opens. Accept the defaults and click **OK**.

Creating the Site Model

Modifying Objects by Record Value

AL Adding Source Data with 2D Polygons

Another way to generate the site model source data is to draw, trace, or import 2D polygons representing contours; then use the **2D Polys to 3D Contours** command to convert the polygons to 3D polygons.

The **Modify by Record** command can convert 2D polygons or polylines with attached record elevation data directly to 3D contours at the proper elevation. See "Modifying Objects by Record Value" on page 270.

To generate source data with the **2D Polys to 3D Contours** command:

1. Create or import the 2D polygons representing contour lines.

   Polylines cannot be used. If contours are drawn with polylines, convert them to polygons first with the **Modify > Convert > Convert to Polygons** command.

2. Select the **2D Polys to 3D Contours** from the appropriate menu:

   - Landmark workspace: **Landmark > Survey Input > 2D Polys to 3D Contours**
   - Architect workspace: **AEC > Survey Input > 2D Polys to 3D Contours**

   The 2D Poly to 3D Contour Setup dialog box opens. Enter the first elevation and change in elevation between contour lines, and then select the type of object to be created.

Click to show/hide the parameters.

Parameter	Description
Start Elevation	Indicates the elevation of the lowest contour in the drawing
Interval	Specifies the change in elevation from one contour line to the next
Create 3D Polygons/3D Loci	Select whether to create 3D polygons or 3D loci

3. Click **OK**.

   The Set Elevation dialog box opens, and the first polygon in the drawing order is highlighted. If necessary, use the **Up** or **Dn** buttons to set the elevation of the highlighted polygon, and then click **Next**. When prompted, indicate whether to delete the original 2D polygons.

Click to show/hide the parameters.

Parameter	Description
Up/Dn	Changes the elevation of the selected polygon
Use same height as prev.	Sets the elevation for successive polygons to the same value
Next	Selects the next polygon
Done	Stops setting polygon elevations; click when all the polygon elevations have been specified

4. Repeat for all of the polygons in the drawing.

   To interrupt this process, click **Done**. To resume setting elevations, select the starting polygon and select the **2D Polys to 3D Contours** command.

Either 3D polygons or loci are created from the 2D polygons, with Z values equal to the assigned elevations. These Z values can be edited in the Object Info Palette if they are incorrect.

Validating 3D Source Data
Simplifying 3D Polygons
Creating the Site Model

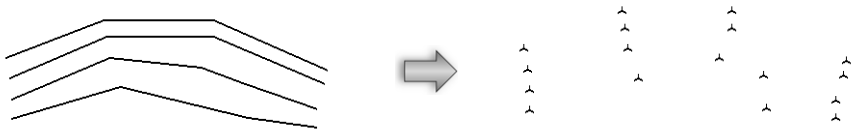
L Converting 3D Polygons to 3D Loci

The **3D Polys to 3D Loci** command converts the vertices of 3D polygons into 3D loci. These loci can then be used to create the site model.

To create 3D loci from 3D polygons:

1. Select the **3D Polys to 3D Loci** command from the appropriate menu:
   - Designer workspace: **AEC > Survey Input > 3D Polys to 3D Loci**
   - Landmark workspace: **Landmark > Survey Input > 3D Polys to 3D Loci**

   The 3D Poly to 3D Loci Command dialog box opens.
2. To remove the 3D polygons, select **Delete original 3D polygons**.
3. Click **OK**. The vertices of any 3D polygons located in the current layer are converted to 3D loci. The loci retain the Z values of the original polygons. These 3D loci can then serve as source data for the site model, or can be converted to stake objects and modify an existing site model.



Original 3D polygons Resulting 3D loci

Validating 3D Source Data
Creating the Site Model
Converting 3D Loci to Stake Objects

A L Adding Source Data by Grid Method Entry

The **Grid Method Entry** command creates a rectangular grid of points on the screen, and prompts you to enter the elevations of those points. The **Create Site Model** command then creates a topographical model based on those points.

Use this method when the source data is extracted from a paper map. Draw a grid on the map, and then determine the elevation of each grid intersection by interpolation between adjacent contour lines. Use these elevation values to create the grid.

To generate source data using grid method entry:

1. Select the **Grid Method Entry** command from the appropriate menu:
   - Architect workspace: **AEC > Survey Input > Grid Method Entry**
   - Landmark workspace: **Landmark > Survey Input > Grid Method Entry**

   When prompted, click at the upper left corner of the grid.
2. After the starting point is picked, the Point Entry Grid Setup dialog box opens.

   Enter the number of rows and columns of points to be created, as well as the distance between the points (**Grid Spacing**), and then click **OK**. A grid of red 3D loci is created.

3. The Enter Z Values dialog box opens.

   The first point on the drawing, at the upper left hand corner, is automatically selected. Enter the elevation of the point, and click **Next**. The elevation is applied to that point, and then the second point in that row is automatically selected. Enter the elevation for that point; continue until elevations have been entered for all of the points in the grid, and then click **Done**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Validating 3D Source Data
Simplifying 3D Polygons
Creating the Site Model

Validating 3D Source Data

Use the **Validate 3D Data** command before you create the site model, to check the 3D source data for errors. Even if a site model already exists, the source data can be checked. If problems occur due to erroneous site model source data, error alerts are automatically displayed. A site model with errors can still be created; however, results may not be as expected.

The following problems can generate errors:

- Duplicate 3D data points or polygons
- 3D data points that are coincident or vertically placed
- Crossing contours (3D polygons) in the 3D source data

To validate 3D source data:

1. Select the source data (3D loci, 3D polygons, or stake objects) to be checked.
2. Select the **Validate 3D Data** command from the appropriate menu:
   - Architect workspace: **AEC > Terrain >Validate 3D Data**
   - Landmark workspace: **Landmark > Validate 3D Data**
3. If the source data are valid, a dialog box states that no problems were found. Proceed with creating the site model as described in “Creating the Site Model” on page 694.
4. If there are errors in the source data, the Problems with Site Model Source Data dialog box opens.

   Problems encountered are listed, and some conditions have an associated button for correcting each error by modifying or deleting the erroneous data.

   If a site model already exists, the problem data can be viewed by selecting the site model and selecting **Modify > Edit Site Model**, or right-clicking (Windows) or Ctrl-clicking (Mac) and selecting **Edit Site Model Source Data** from the context menu. Problem data conditions are highlighted and annotated.
5. When problem correction is complete, click **Close**.

   Select **Validate 3D Data** again to ensure that the errors have been corrected.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Simplifying 3D Polygons
Correcting Site Modifier Errors

Simplifying 3D Polygons

If the existing site model was created from 3D polygons (not 3D loci), you can reduce processing time by removing redundant polygon vertices within a specified tolerance. For example, this removes extra vertices in straight or nearly-straight segments of imported 3D polygons used as source data. This feature does not significantly change the appearance of the site model, but it can dramatically decrease the time it takes to process the model.

The **Simplify 3D Polygons** command available in Vectorworks Architect and Landmark is specialized for certain site model workflows. To simplify 2D polygons, 3D polygons, and polylines in other workflows, see "Simplifying Polygons and Polylines" on page 1090.

To remove redundant 3D polygon vertices:

1. Select the 3D polygon(s). (The original polygons remain unchanged.)
2. Select the **Simplify 3D Polygons** command from the appropriate menu:
   - Architect workspace: **AEC > Terrain > Simplify 3D Polygons**
   - Landmark workspace: **Landmark > Simplify 3D Polygons**

   The Simplify Selected 3D Polygons dialog box opens.
3. Enter a **Simplification Tolerance** value, and select a layer from the **Put Simplified Result in Layer** list.
4. Click **OK**.

   Redundant vertices are removed from the 3D polygon(s) and the results are sent to the layer specified.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Validating 3D Source Data
Simplifying Polygons and Polylines

A L Creating the Site Model

Creating a Site Model from the Source Data

Once source data has been obtained for the site model, the site model can be created. After generation, the site model is cached so that updating occurs more efficiently; however, large file sizes can result. See "Document Display Preferences" on page 60.

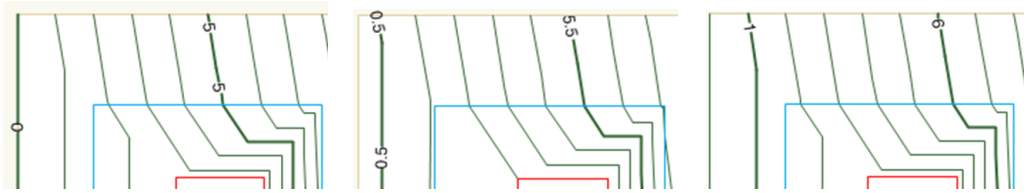
To create a site model from source data:

1. Ensure that valid source data exists. The site model can be created from 3D loci, 3D polygons, stake objects, or by one of the methods described in "Site Model Source Data" on page 689. Verify that there are no problems with the source data with the **Validate 3D Data** command (see "Validating 3D Source Data" on page 693).
2. Select the source data.
3. Select the **Create Site Model** command from the appropriate menu:
   - Architect workspace: **AEC > Terrain > Create Site Model**
   - Landmark workspace: **Landmark > Create Site Model**

   The Site Model Settings dialog box opens. Specify the site model parameters.

Click to show/hide the parameters.

Parameter	Description
Site Model Name	Enter a name for the site model object
Keep Original Source Data in Layer	Leaves a copy of the source data in the layer (the site model always retains a copy of the source data in its profile group). This parameter displays only during initial site model creation.
Settings	
Minor Contour Interval	Specifies the standard contour line placement interval
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Major Contour Multiple	Specifies the placement interval of major (heavy) contour lines
Start Contour Offset	Controls the starting elevation of the 2D contours  Start Contour Offset = 0 Minor Contour Interval = 1 Major Contour Interval = 5 Start Contour Offset = 0.5 Minor Contour Interval = 1 Major Contour Interval = 5 Start Contour Offset = 1 Minor Contour Interval = 1 Major Contour Interval = 5
Maximum Elevation	Specifies the highest contour elevation to be displayed. No contours are created if the maximum elevation is lower than the minimum elevation, but this can be useful for cut and fill calculations (for example, for pond or backfill volumes). If a site model update will result in contours exceeding the minimum/maximum range, an alert displays, allowing the range to be adjusted.
Minimum Elevation	Specifies the lowest contour elevation to be displayed
Datum Elevation	Indicates the reference elevation used to define the bottom of a site model when the 3D Mesh Solid style is selected in **3D Display Settings**; set to zero for normal cut and fill calculations
Flow Arrow Spacing	Determines the placement of the flow arrows
3D Grid Spacing	Sets the grid (mesh) size for 3D display; also determines flow arrow placement
Smoothing Interval	When selecting a triangulation method for 2D and/or 3D site model display, sets the distance for adding new points to the contours. Certain site models, since they are created from randomly-located origin points, may have an uneven triangulation distribution. Smoothing forces the site model to create an even distribution of data points along each contour line, simulating even sampling. Enter zero to use the original contour points for triangulation. Low values may result in long calculation times.
2D Display Settings	
Display	Select whether to display the existing site model only, proposed site model only, or both existing and proposed site models. The display option can be changed later from the Object Info palette.
Existing Only	Displays only the existing model, including any site modifiers that apply to the existing site model
Proposed Only	Displays only the proposed model (the current model changed by any site modifiers that apply to the proposed site model)
Proposed + Existing	Displays both the proposed model and the existing model; this option is useful when using the site model for a grading plan, with different existing and proposed topographical line styles
Style	Select a 2D site model display mode
2D Contour	Draws contour lines
2D Contour (smoothed)	Smooths contour lines

Parameter	Description
2D Contour (colored elevations)	Draws contour lines and colors the elevations for conducting an elevation analysis (elevation analysis parameters are specified by clicking **Graphic Properties** and selecting the Site Analysis tab)
2D Contour (colored slopes)	Draws 2D contour lines by triangulation, according to the specified **Smoothing Interval**, and colors the slopes within specified angle ranges for conducting a slope analysis
2D Triangle	Represents the surface as triangular facets
2D Triangle (colored slopes)	For 2D triangle display, colors the slopes within specified angle ranges for conducting a slope analysis (slope analysis parameters are specified by clicking **Graphic Properties** and selecting the Site Analysis tab)
2D Triangulated Contours	Draws 2D contour lines by triangulation, according to the specified **Smoothing Interval**
Show Flow Arrows	Adds arrows to show the downward direction for drainage analysis
Show Labels	Labels major 2D contour line elevations
Parallel to the topo line	When labels are selected for display, labels are parallel to the topography lines
Draw Site Border	Draws a closed 2D polygon along the site border (outer edge) of the site model
3D Display Settings	
Display	Select the 3D site model display. The display option can be changed later from the Object Info palette.
No 3D Display	Displays only the 2D site model, with no 3D component
Existing Only	Displays the existing 3D site model
Proposed Only	Displays the proposed 3D site model
Cut and Fill	Displays the 3D cut and fill volumes. Areas where existing and proposed site models are the same are represented by a flat area. Cut areas display as a depression in the 3D surface, and fill areas display as a raised area in the 3D surface. (Select cut and fill colors by clicking **Graphic Properties** and selecting the Site Model tab.)
Style	Select a 3D site model display mode
3D Contour	Draws the site model using horizontal 3D polygons
3D Grid	Draws the site model using a regular grid (mesh) of horizontal four point rectangles, spaced by the **3D Grid Spacing** value; the mesh can be colored, filled, and rendered with shadows
3D Mesh solid	Groups 3D triangles to form a Vectorworks mesh; a skirt and bottom are added to create a valid solid
3D Triangle	Generates 3D triangles with the Triangulated Irregular Network (TIN) method; this method is the most accurate, as it involves no interpolation
3D Triangles (colored slopes)	Generates 3D triangles with the Triangulated Irregular Network (TIN) method, and colors the slopes within specified angle ranges for conducting a slope analysis
3D Extruded Contour	Creates horizontal solids with a thickness equal to the contour interval; top edges run along contour lines or the site border. This creates a “layer cake” or “chipboard model” effect.

Parameter	Description
3D Triangulated Contours	Draws 3D contour lines by triangulation, according to the specified **Smoothing Interval**
3D Triangulated Contours (colored slopes)	Draws 3D contour lines by triangulation, according to the specified **Smoothing Interval**, and colors the slopes within specified angle ranges for conducting a slope analysis
Show 3D Grid	Superimposes a 3D grid on the model; specify the grid spacing value in **Spacing**
Fill	Displays 3D grid squares with a fill; **Show 3D Grid** does not need to be selected to display a filled grid
Show 3D Contour	Superimposes a 3D contour on the model
Use Site Modifiers on	
All Layers	Modifies the proposed site model with modifiers from any layer in the file
Visible Layers Only	Modifies the proposed site model with modifiers from visible layers only; modifiers on invisible layers do not affect the site model
Same Layer as Site Model only	Modifies the proposed site model with modifiers that exist only on the site model layer
Graphic Properties	Specifies the site model appearance, along with slope and elevation analysis parameters

4. Click **OK**.

   A site model is created in the active layer and displays as specified.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Site Model Source Data
Setting Site Model Graphic Properties
Site Model Properties
Creating a Site Model Snapshot

Setting Site Model Graphic Properties

The site model 2D and 3D graphic display properties can be specified to obtain the desired graphic component colors and line styles. In addition, specify the colors for elevation and slope analyses, and for the 3D cut and fill volume representation. These graphic properties can be set when the site model is first created, or changed later from the Object Info palette.

To set the site model graphic properties:

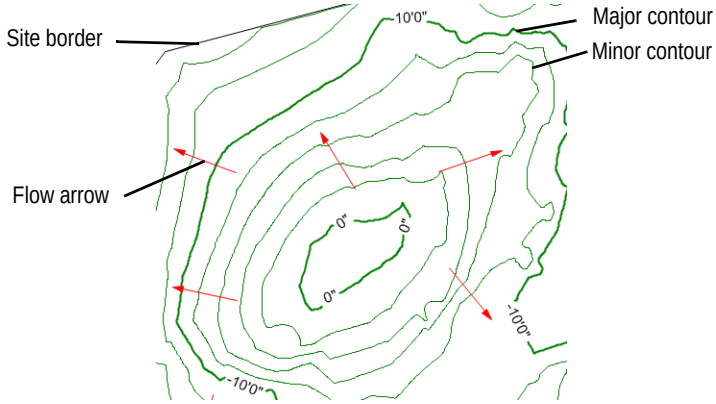
1. When creating a new site model, click **Graphic Properties** from the Create Site Model dialog box. To edit a current site model, click **Site Model Settings** from the Object Info palette, double-click the site model, or right-click (Windows) or Ctrl-click (Mac) on a site model and select **Edit** from the context menu to open the Site Model Settings dialog box, and then click **Graphic Properties** from the Site Model Settings dialog box.

   The Graphic Properties dialog box opens.

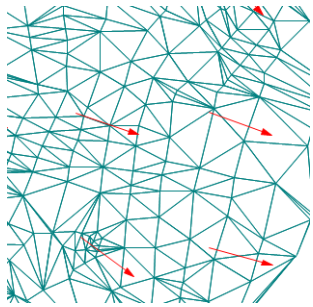
2. On the Site Model tab, the 2D (existing and proposed) and 3D site model display can be completely customized by selecting the class and color for all components, as well as the line type, line thickness, and, for certain components, marker style. Double-click on a line to set attributes, or select the item and click **Edit**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

An Attributes dialog box opens. Select an overall class for each site model component to control its appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, or select the class named <Site Model Class> which places the site model component in the same class as the site model object.

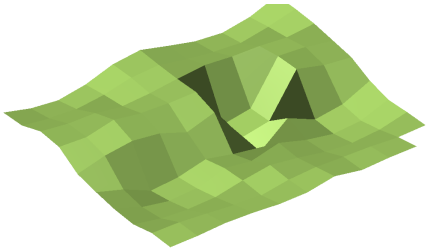
Each attribute (color, line type, line thickness, and marker) can be set to the "By Class "setting. The attributes are then controlled by the class assigned to the component. (See "Setting Class Attributes" on page 181.)



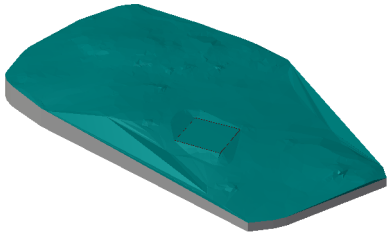
2D Contour style



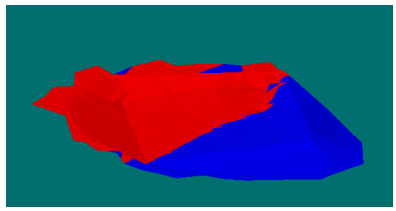
2D Triangle style



3D Grid style



3D Mesh Solid style



Cut and Fill volumes

3. On the Site Analysis tab, set the slope and elevation analysis graphic parameters. A slope analysis shows the slope change of the site model with color ranges indicating the steepest to the shallowest slope. An elevation analysis shows the elevation change of the site model as a gradient of color from the minimum to the maximum elevation.

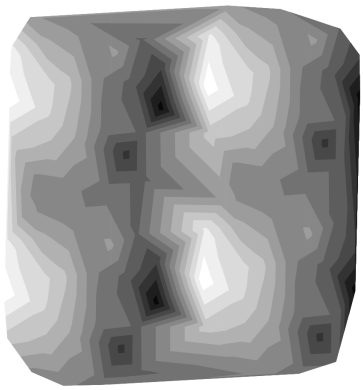
Click to show/hide the parameters.

Parameter	Description
Slope Colors	Specify the slope analysis settings
Number of slope categories	Indicate the number of slope levels to use in the analysis
Category __ maximum slope %	For each category, indicate the maximum slope percentage and its associated color; slopes below the specified percentage will be colored as specified
Slopes exceeding maximum category	Slopes above the maximum category's percentage will receive the selected color
Elevation Colors	Specify the elevation analysis color settings; a color gradient from the minimum to the maximum elevation is applied to the site model for the elevation analysis
Minimum Elevation Color	Select the starting color for the lowest elevation

Parameter	Description
Maximum Elevation Color	Select the starting color for the highest elevation
2D Cut & Fill Area	Draws a polygonal representation of the cut and fill over the 2D representation of the site model with the specified colors
Cut Volume Area Color	Select the color for the volume that is cut from the site model
Fill Volume	Select the color for the volume of fill required for the site model

4. Click **OK** to close the Graphic Properties dialog box.
5. To display a slope analysis, select a 2D display style of 2D or 3D Triangle (colored slopes), 2D Contour (colored slopes), or 3D Triangulated Contours (colored slopes) for the site model. To display an elevation analysis, select a 2D display style of 2D Contour (colored elevations) for the site model.

Slope analysis



Elevation analysis

Create a snapshot of the site model to easily display a slope or elevation analysis along with the site model (see “Creating a Site Model Snapshot” on page 704).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Site Model from the Source Data
Site Model Properties
Cropping a Site Model
Creating a Site Model Snapshot
Sending Objects to the Site Model Surface

AL Setting Site Model Properties

Site Model Properties

The site model object can be copied, cut, pasted, rotated, and deleted like other objects. In 2D Top/Plan view, the **2D Display** selected for the site model is shown; if both existing and proposed models are selected for display, they are superimposed. In any 3D view, the **3D Display** selected (if any) is shown.

If the Renderworks product is installed, the site model can be textured from the Render tab of the Object Info palette. The texture is applied to the surface of the site model, and is scaled to fit the site model bounding box.

To see the original source data that were used to create the site model, select **Modify > Edit Site Model**, or select **Edit Site Model Source Data** from the site model context menu. Changes can be made to the source data, and the source data can be validated (see “Validating 3D Source Data” on page 693). Return to the site model by clicking **Exit Source**
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Data at the top right of the drawing window. Click **Update** in the Object Info palette, or select **Update** from the site model context menu, to reflect any source data changes in the site model.

Select the **Create Drape Surface** command to create a "smoothed" version of the site model contours (see "Creating a Drape Surface" on page 337).

Once generated, the site model is cached so that updating occurs more efficiently; however, large file sizes can result. See "Document Display Preferences" on page 60.

In addition to the Site Model Settings dialog box, you can change many parameters of a selected site model from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Site Model Settings	Opens the Site Model Settings dialog box. The settings here (and the related graphic properties settings) are identical to those specified at site model creation. The option to keep or delete original source data is only available at site model creation.
Update	Updates the site model and any associated snapshots, when the existing site data has changed or site modifiers have been added or changed
Create a Snapshot	Creates a snapshot of the site model with the current site model settings, for displaying different versions of the site model
2D Display	Select the display type for the 2D site model
2D Style	Select the display style for the 2D site model
3D Display	Select the display type (or no display) for the 3D site model
3D Style	Select the display type for the 3D site model
Area Display Type	Select the units for displaying the projected and surface areas for existing and proposed site models
Projected Area	Displays the projected area (the area projected in plan view) of the site model
Surface Area (Existing)	Displays the 3D surface area of the existing site model
Surface Area (Proposed)	Displays the 3D surface area of the proposed site model (when one exists)
Update Cut and Fill Calculations	Recalculates the site model cut and fill; after modifying and updating the proposed site model, click to view the resulting cut and fill volumes (see "Creating a Spoil Pile Area" on page 714). When the calculated volumes indicate that updating is necessary, click to display the latest results.
Volume Display Type	Select the units for displaying the cut and fill volumes (or allows cut and fill volumes to be hidden, if desired)
Volume (Existing)	Displays the total volume of the existing site model
Volume (Proposed)	Displays the total volume of the proposed site model
Cut Volume	When the proposed site model has been modified by a site modifier, displays the volume cut from the site model
Fill Volume	When the proposed site model has been modified, displays the volume of fill added to the site model

Parameter	Description
Net C&F Volume	Displays the net cut or fill volume, indicating whether fill is required or extra cut volume needs to be removed from the site
Total C&F Volume	Displays the sum of the cut and fill volumes
Error Count	When site modifier errors prevent the site model from updating correctly, displays the number of errors in red; see “Correcting Site Modifier Errors” on page 754
Show Errors	Select to display site modifier error icons on the drawing, or deselect to hide the icons

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Updating a Site Model
Cropping a Site Model
Reshaping the Site Model
Editing Contour Labels
Creating a Site Model Snapshot
Sending Objects to the Site Model Surface
Creating the Site Model

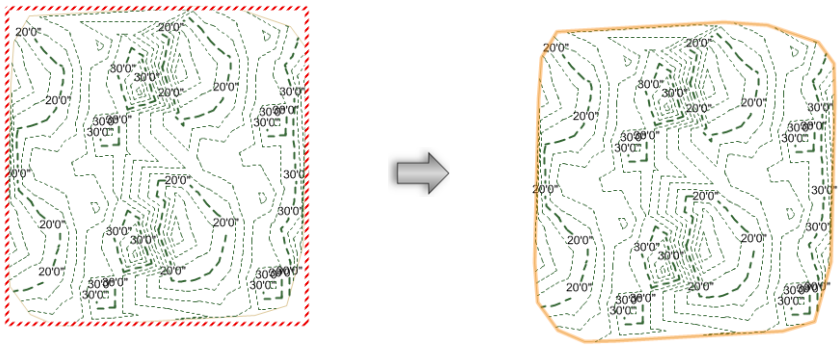
Updating a Site Model

If a site modifier object is moved or modified on any layer, the site model displays as an out-of-date site model with a red and white striped border. This indicates that the site model requires updating.

To update the selected site model:

1. Select the site model.
2. On the Object Info palette, click **Update**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the site model and select **Update** from the context menu.
3. The selected site model is updated.



   If the update causes the site model contours to fall outside the minimum/maximum elevation range specified in the site model settings, an alert displays, allowing the range to be adjusted.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Site Model Properties

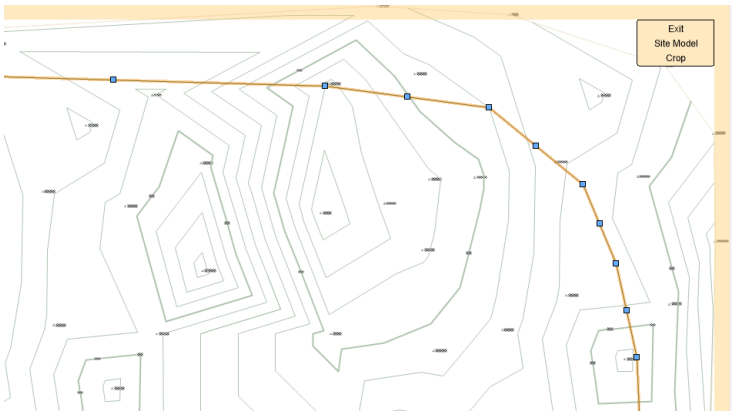
A L Cropping a Site Model

The site model can be cropped by a custom site border shape. This allows you to limit the area of the site model, without permanently excluding the original extents of the source data. The original site model can easily be restored by editing the site model crop object.

To crop a site model:

1. Right-click (Windows) or Ctrl-click (Mac) on the site model and select **Edit Site Model Crop** from the context menu.
2. In site border editing mode, any existing site border object is selected. Move the site border, or use the **Reshape** tool to reshape (edit) the site border object; see “Reshaping Objects” on page 1043. The site border object can also be deleted and a new site border object can be created from any closed 2D shape.

   To remove the cropping from a site model, delete the site border object. This restores the site model to its original bounds.



3. When the site border is ready, click the **Exit Site Model Crop** button (or select **Modify > Exit Site Model Crop**) to return to the drawing.
4. The site border crops the site model object.

   With the **Draw Site Border** option selected in the site model settings, the site border displays in Top/Plan view. Adjust the color of the site border in the graphic properties of the site model settings.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Site Model Properties
Creating a Site Model Snapshot
Reshaping the Site Model
Creating the Site Model

A L Reshaping the Site Model

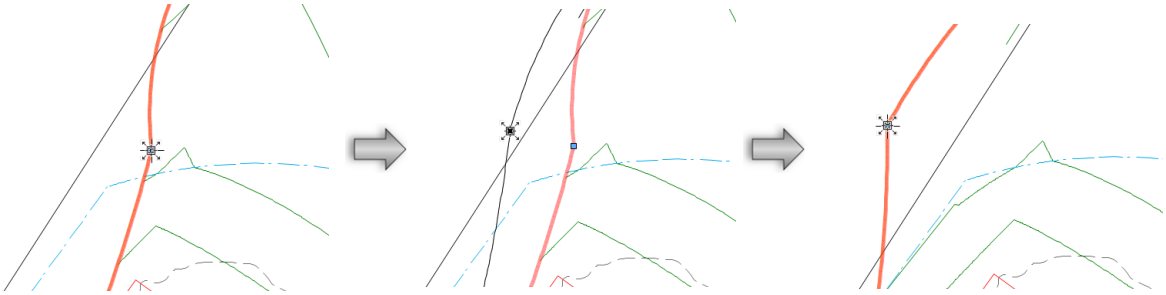
The edge of a selected site model displays a crop or border (select **Draw Site Border** in the Site Model Settings to draw a 2D polygon on the edge of the site model in Top/Plan view). In Vectorworks Architect or Landmark, the **Reshape** tool reshapes the crop. This is similar to editing the site model crop object as described in “Cropping a Site Model” on page 702.

 To reshape the site model crop:

1. Select the site model.
2. Click the **Reshape** tool from the Basic palette, and click **Edit Site Model Crop** from the Tool bar.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Reshape the site model crop as described in “Reshaping Objects” on page 1043. Crop vertices can be repositioned, added, and deleted.

   The site model crop is immediately reshaped. No updating is necessary.



Use the Move Polygon Handles mode of the Reshape tool in Edit Site Model Crop mode to move the site model crop vertex, reshaping the crop

Cropping a Site Model
Site Model Properties

Editing Contour Labels

To change the font size of the contour labels, set the size from the **Text > Size** menu, and then click **Update** in the Object Info palette of a selected site model.

Normally, contour labels are distributed along the major contour lines at regular intervals. Occasionally, this is not desirable and a label needs to be moved. When a site model is selected, two additional modes are available for the **Reshape** tool. These modes allow the contour labels to be repositioned, added, and removed.

Show labels must be selected in the Site Model Settings dialog box.

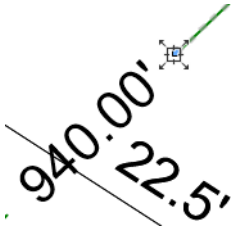
To edit contour labels:

1. Select the site model.
2. Click the **Reshape** tool from the Basic palette, and click **Edit Site Model Label Position** from the Tool bar.

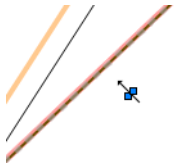
   Each label on the major contours of the site model displays with a control point.

By default, this contour label was placed in a crowded area of the drawing

3. Use the Move Label Handle mode to move the label along the current contour line. Add Label mode adds a label along the contour line; click to add a label at that location. Finally, click on a label control point in Delete Label mode to remove the contour label.



Move Label Handle mode moves the contour label along the contour line; click to place it in its new location



Click on a contour line in Add Label mode to place a new label at that location

Click on a label control point in Delete Label mode to remove the label

Only one contour label can be edited at a time. The **Reshape** tool marquee modes have no effect on contour label operations. See “Reshaping Objects” on page 1043 for more information.

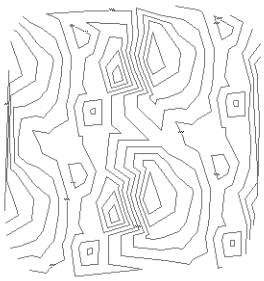
If the site model changes later, the relative label positions are preserved, reducing the need to reposition them again.

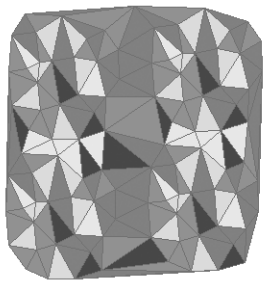
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

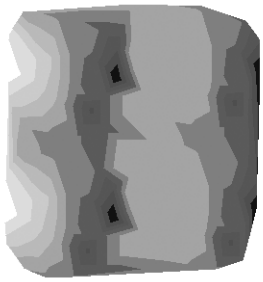
Site Model Properties
Creating a Site Model Snapshot
Reshaping the Site Model
Creating the Site Model

A L Creating a Site Model Snapshot

Creating a snapshot of the site model allows several display styles of the site model to be viewed at one time. The snapshot views cannot be edited directly; however, if the original site model is modified, the changes apply to all related snapshots automatically upon updating.







To create a site model snapshot:

1. Set the properties of the site model as desired for the snapshot. The snapshot’s appearance is based on the current site model settings, and its location is based on the original source data.
2. From the Object Info palette of a selected site model, click **Create a Snapshot**.
3. The snapshot is created on the active layer, over the original site model. The Object Info palette of a selected snapshot displays the parameters of the original site model, and cannot be changed for the snapshot (with the exception of the display units).
4. Any changes to the original site model are reflected in associated snapshots when clicking **Update** in the Object Info palette of the selected original site model.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Site Model from the Source Data
Updating a Site Model
Site Model Properties

Sending Objects to the Site Model Surface

Trees, parking areas, retaining walls, site retaining modifiers, and other site objects may need to be placed on the surface of the model. For this purpose, use the **Send to Surface** command.

This command is effective for 3D loci, 3D polygons, extrudes, sweeps, meshes, slabs, light objects, walls, and symbols with a 3D component. If a 2D polyline or 2D polygon is selected, it will first be converted into its 3D equivalent before being sent to the surface.

To send an object to the surface:

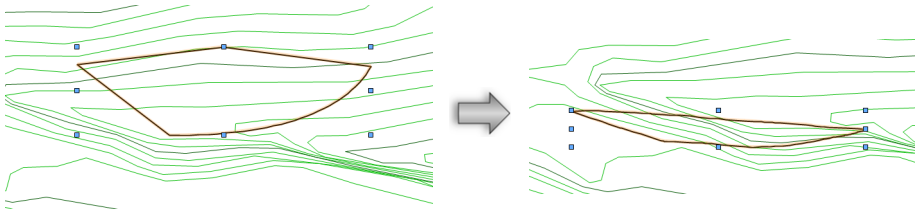
1. Make the layer with the site model object the active layer.
2. Place the object on the active layer plane and move it to the desired location on the site model.
3. Select the **Send to Surface** command from the appropriate menu:
   - Architect workspace: **AEC > Terrain > Send to Surface**
   - Landmark workspace: **Landmark > Send to Surface**

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object and select **Send to Surface** from the context menu (Vectorworks Landmark required).
4. If the object is a retaining wall or pad with retaining edge site modifier, the Send to Surface dialog box opens. Select the method of sending the object to the surface of the site model, and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Elevate the retaining edge	Sets the existing vertices of the retaining edge or retaining wall left/right edge to the surface elevation
Fit the retaining edge	Fits the retaining edge or retaining wall left/right edge to the surface of the site model, modifying existing vertices and/or adding new vertices
Elevate the pad	Sets the elevation of the retaining edge pad or retaining wall pad to the surface elevation; the edges remain unchanged

5. The correct Z value is given to the object's insertion point, bottom-most point, or specified portion, placing the object on the surface of the site model.



Creating a Site Model from the Source Data
Site Model Properties

Site Model Modification Overview

As discussed in "Sitework Overview" on page 689, when the site model is generated, two different display modes of the site model object are available—existing and proposed. The existing site represents the site prior to your project; the proposed site incorporates the project's changes. To modify the site model, use the **Site Modifiers** tool, along with

other site modification tools in the Vectorworks program; a site modifier can be applied to either the existing or proposed site model.

Site modifiers can be placed on any layer. The site model properties specify whether modifiers in another layer can affect the site model (see “Creating the Site Model” on page 694). You might put various site modifiers on separate layers, for example, so that you can show each modifier’s effect on the site model individually. When a site modifier is placed in the drawing, the Site-DTM-Modifier class is created automatically.

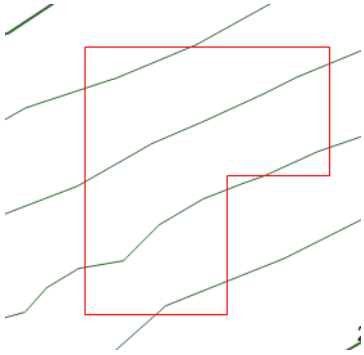
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Pad

A pad is a 3D polygon representing the shape of an element which is to be added to, and normally, modifies, the site model. The modifier can be applied to the existing or proposed site model; the site model is modified when the site model is updated. Draw a pad with the **Site Modifiers** tool, or draw a polyline and then select the **Create Objects from Shapes** command to change it to a pad object (see “Creating Objects from Shapes” on page 277).

To create a pad:

1. Click the **Site Modifiers** tool from the Site Planning tool set.
2. Draw the pad object. Pads can be drawn closed or open (open pads are sometimes called “break lines”).



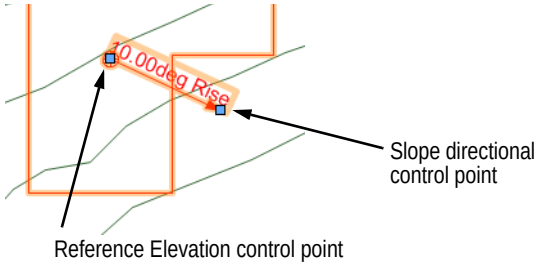
If this is the first time the **Site Modifiers** tool has been used, the Object Properties dialog box opens. Accept the defaults and click **OK**.

3. In the Object Info palette, select **Pad** in the **Config** field.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

4. If the pad is sloping, enter the **Slope** value (positive for rising, negative for falling), and select the slope definition unit (**Angle** or **Percent** in the **SlopeDef** field).

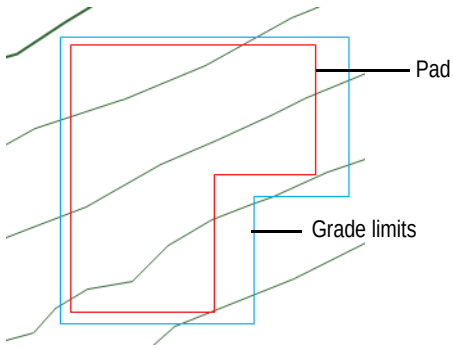
   A slope arrow is automatically drawn on the pad, indicating the direction of the slope. The bull's-eye at the start of the slope arrow indicates the zero elevation point of the sloping pad. Select the control points of the slope arrow to move either end. Deselect **Show Slope Arrow** to hide the slope arrow.

Click to show/hide the parameters.

Parameter	Description
Config	Indicates the type of modifier object
Elevation	Sets the reference elevation of the pad at its control point; if a Z-height is also specified, the resulting elevation is additive
Slope	Determines the angle of the pad; if the value is zero, the pad has no slope, and the following two parameters are not displayed
SlopeDef	Select **Angle** or **Percent** to determine the slope units
Apply To	Specifies whether the site modifier applies to the existing or proposed site model
Show Slope Arrow	If selected, displays a modifiable slope arrow and the current slope in Top/Plan view; **Show Slope Arrow** only displays if the slope is a value other than zero 
Update Calculations	Updates the area and volume calculations displayed in the Object Info palette for the area of the site model located under the site modifier
Site model area and volume data	Displays the area and volume information for the site model area located under the site modifier; select the units for the area and volume. For more information, see "Site Model Properties" on page 699.

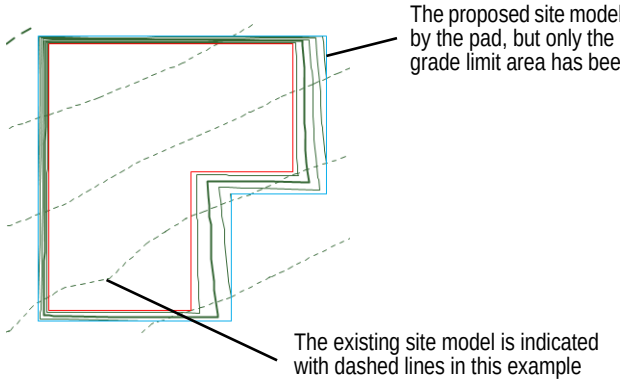
The **Site Modifiers** tool generates a 3D pad polygon modifier that is always planar.

5. Normally, create grade limits around the pad.

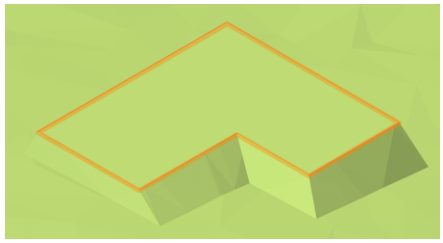


Site modifiers before updating the site model

6. The site model requires updating to reflect the pad modification. Select the site model and click **Update** from the Object Info palette.



2D view



Rendered 3D view

For predictable results, pads should not overlap. All pads should be either completely inside or completely outside of all grade limits. Pads, including those that may be incorporated into plug-in objects (such as landscape walls) cannot cross outside the grade limits.

Creating a Pad with a Retaining Edge
Creating Grade Limits
Creating Grade Limits Automatically
Creating Custom Site Modifiers
Updating a Site Model

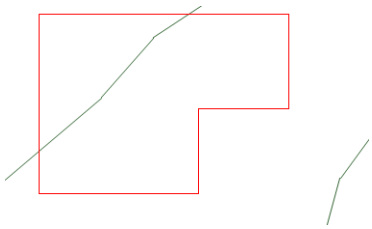
Creating a Pad with a Retaining Edge

A pad with a retaining edge is similar to a regular pad site modifier, but it includes an additional configuration that defines an edge around the pad, allowing the terrain to be shaped around the pad. For example, the retaining edge can be used to adjust the terrain around a hardscape, offering precise control over the cut and fill amounts.

The modifier can be applied to the existing or proposed site model; the site model is modified when the site model is updated. Draw a pad with the **Site Modifiers** tool, or draw a polyline and then select the **Create Objects from Shapes** command to change it to a pad object (see “Creating Objects from Shapes” on page 277).

To create a pad with a retaining edge:

1. Click the **Site Modifiers** tool from the Site Planning tool set.
2. Draw the pad object; the retaining edge is added automatically.



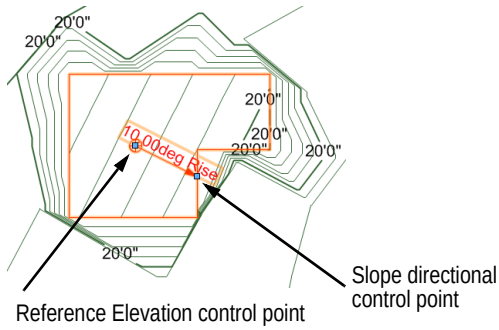
If this is the first time the **Site Modifiers** tool has been used, the Object Properties dialog box opens. Accept the defaults and click **OK**.

3. In the Object Info palette, select **Pad with Retaining Edge** in the **Config** field.

4. If the pad is sloping, enter the **Slope** value (positive for rising, negative for falling), and select the slope definition unit (**Angle** or **Percent** in the **SlopeDef** field).

   A slope arrow is automatically drawn on the pad, indicating the direction of the slope. The bull's-eye at the start of the slope arrow indicates the zero elevation point of the sloping pad. Select the control points of the slope arrow to move either end. Deselect **Show Slope Arrow** to hide the slope arrow.

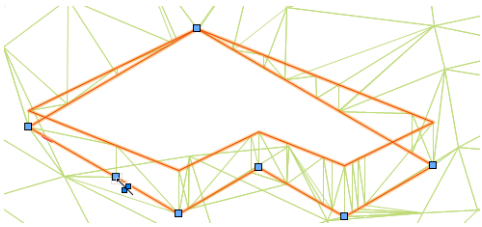
Click to show/hide the parameters.

Parameter	Description
Config	Indicates the type of modifier object
Elevation	Sets the reference elevation of the pad at its control point; if a Z-height is also specified, the resulting elevation is additive
Slope	Determines the angle of the pad; if the value is zero, the pad has no slope, and the following two parameters are not displayed
SlopeDef	Select **Angle** or **Percent** to determine the slope units
Apply To	Specifies whether the site modifier applies to the existing or proposed site model
Show Slope Arrow	If selected, displays a modifiable slope arrow and the current slope in Top/Plan view; **Show Slope Arrow** only displays if the slope value is other than zero 
Update Calculations	Updates the area and volume calculations displayed in the Object Info palette for the area of the site model located under the site modifier
Site model area and volume data	Displays the area and volume information for the site model area located under the site modifier; select the units for the area and volume. For more information, see "Site Model Properties" on page 699.
Vertex parameters	Edits the path vertices; see "Editing Vertex-Based Objects" on page 1002
Move	Select the portion of the retaining edge modifier to edit (vertex or edge), and then edit its **Z** value, or scroll through its vertices with the left and right arrows and change the selected vertex **Z** value
Edit	For vertex selection made in **Move**, scrolls through the vertices, highlighting the currently selected vertex. Click the center button to highlight the selected vertex.
Z	Sets the elevation of the modifier item selected in **Move** or the vertex selected in **Edit**

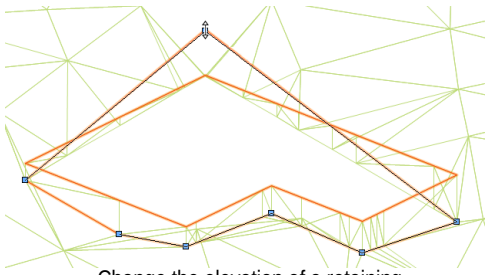
The **Site Modifiers** tool generates a 3D pad polygon modifier that is always planar.

5. The retaining edge modifier is drawn separately, and it can be used to control the depression around the pad. Reshape the retaining edge either from the Object Info palette or with the **Reshape** tool.

- The Object Info palette vertex parameters include the ability to edit the Z value of the selected retaining modifier vertex or retaining modifier edge.
- The retaining edge site modifier can be reshaped with the **Reshape** tool, similar to reshaping walls (see “Reshaping Walls” on page 536). Move vertices, and add or delete vertices to reshape the retaining edge.



Add a vertex to the retaining edge with the Reshape tool



Change the elevation of a retaining edge vertex with the Reshape tool

- The **Send to Surface** command can be used to send either the retaining modifier edge or the pad to the surface of the site model.

6. The site model requires updating to reflect the pad modification. Select the site model and click **Update** from the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Pad
Creating Grade Limits
Creating Grade Limits Automatically
Creating Custom Site Modifiers
Sending Objects to the Site Model Surface
Updating a Site Model
Site Model Properties
Creating Objects from Shapes

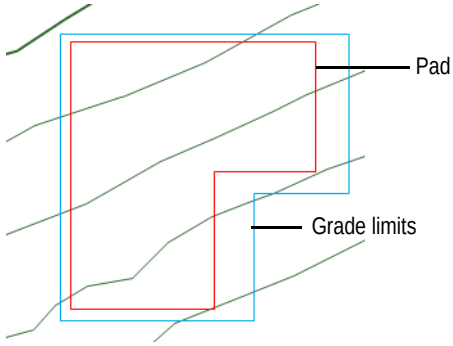
Creating Grade Limits

Grade limits define the area of topographical transition between existing site data and the site modifiers enclosed within the grade limits; they can be thought of as a “limit of construction.” Outside the grade limits, the site remains unchanged; within the grade limits, the site model is defined by the pads. Grade limits can overlap, but cannot completely enclose, another grade limit area. Grade limits should not cross a pad or spoil pile.

Grade limits are created automatically around certain site modifiers, such as massing models. Grade limits can also be created automatically from pads (Vectorworks Landmark required). To draw grade limits manually, use the **Site Modifiers** tool, or draw a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

To create grade limits:

1. Click the **Site Modifiers** tool from the Site Planning tool set.
2. Draw the grade limit area around a site modifier, such as a pad.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Site modifiers before updating the site model

If this is the first time the **Site Modifiers** tool has been used, the Object Properties dialog box opens. Accept the defaults and click **OK**.

3. In the Object Info palette, select **Grade Limits** from the **Config** field.

Click to show/hide the parameters.

Parameter	Description
Config	Indicates the type of modifier object
Apply To	Specify whether the grade limits apply to the existing or proposed site model
Show 3D	In 3D views, represents the grade limits with a 3D polygon; deselect this option to hide this geometry
Update Calculations	Updates the area and volume calculations displayed in the Object Info palette for the area of the site model located under the site modifier
Site model area and volume data	Displays the area and volume information for the site model area located under the site modifier; select the units for the area and volume. For more information, see “Site Model Properties” on page 699.

4. The site model requires updating to reflect the grade limit modification. Select the site model and click **Update** from the Object Info palette.

Creating Grade Limits Automatically
Creating Custom Site Modifiers
Updating a Site Model

Creating Grade Limits Automatically

Because grade limits are often associated with a pad or other type of site modifier, the Vectorworks Landmark product can create grade limits automatically around a site modifier, based on specific parameters. Grade limits can be created with a gradual sloping transition from the existing site contours to the modifier, placed at a variable or fixed distance from the modifier. The grade limits cut and fill the site model to create a constant slope (batter slope).

To create grade limits automatically around one or more site modifier(s):

1. Select the site modifier(s) which require the grade limits. Valid modifiers include pads, road objects, massing models, and landscape walls.
2. Select the **Create Grade Limits from Pad** command from the appropriate menu:
   - Designer workspace: **AEC > Terrain > Create Grade Limits from Pad**

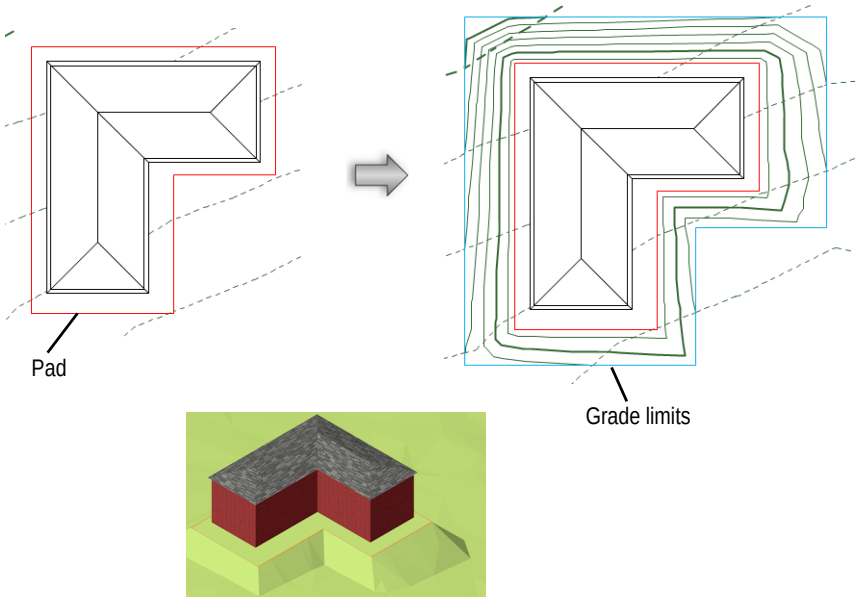
- Landmark workspace: **Landmark > Create Grade Limits from Pad**

The Create Grade Limits from Pad dialog box opens. Select the grade limit parameters.

Click to show/hide the parameters.

Parameter	Description
Grade Limits Offset Control	
Distance	Creates the grade limits at a fixed offset distance
Batter Slope	Creates variable offset grade limits based on slope (the slope variation determines the offset; the grade limits are located at the intersection of the slope and the existing terrain)
Max. Offset	Sets the fixed offset distance for **Distance** offset control, or specifies the maximum offset distance allowed for **Batter Slope** offset control
Max. Slope	For **Batter Slope** offset control, specifies the maximum allowable slope (in degrees)
Slope Control Options	
Use Midpoints	Enables additional testing points for a more uniform slope
Test Increment	Specifies the number of midpoints to test; a smaller increment distance increases the accuracy of the slope calculations, but can also increase the time to create the grade limits (the minimum increment is 12”)

3. Click **OK** to create the grade limits.
4. Select the site model and click **Update** from the Object Info palette.



Updating a Site Model

Creating a Pad from Grade Limits

With Vectorworks Landmark, a pad modifier can automatically be created within defined grade limits. The pad is placed so that the cut and fill amounts of the underlying proposed site model are set equally, helping site designers start with a balanced solution. Batter slope angles can be different on the different sides of the modifier.

Certain restrictions apply when creating the pad:

- The polygon that defines the grade limit area must be closed. If more than one grade limit area is selected, all the grade limit polygons must be closed. (To close the polygon, select the grade limit and then select **Closed** in the Object Info palette.)
- If more than one grade limit is selected, the created pads must have a uniform batter slope. To set the batter slopes individually, select only one grade limit object at a time.
- The grade limit area must be fully enclosed within a site model.
- The specified batter slope or slopes cannot be too small to create the pad; slope values that are too low prevent balanced cut and fill.

To create a pad automatically from the grade limits:

1. Select one or more grade limit objects.
2. Select the **Create Pad from Grade Limits** command from the appropriate menu:
   - Designer workspace: **AEC > Terrain > Create Pad from Grade Limits**
   - Landmark workspace: **Landmark > Create Pad from Grade Limits**

   The Create Pad from Grade Limits dialog box opens, and if necessary, the view temporarily changes to optimally display the area. Specify a uniform batter slope for the entire pad, or scroll with the **Prev** and **Next** buttons to set the slope individually for each side of the pad. The slope value(s) must be greater than 0 but less than 90 degrees.

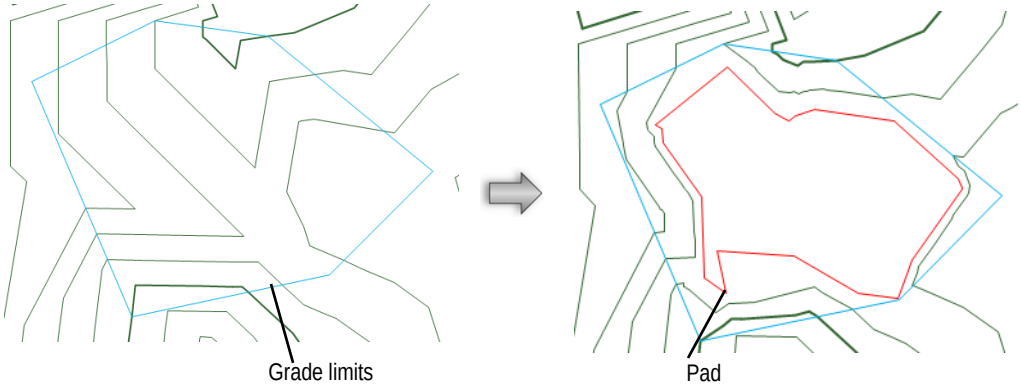
Click to show/hide the parameters.

Parameters	Description
Uniform	Applies the batter slope value equally to all sides of the pad
Per side	Sets a slope value for the selected side of the pad
Select a side	Scrolls through each side of the pad; enter an individual slope value for each selected side
Data display method	Select the method of specifying the slope value: • **Angular**: specifies the slope in degrees, radians, or gradians (example: 26.7°) • **Rise-over-run**: specifies the slope as a ratio of vertical change to horizontal change (example: 1:3) • **Percent**: specifies the slope as a grade percentage (example: 50%)

3. Click **OK**.

   The resulting pad modifier is created and selected.

   Individual slope values are retained for each grade limit object. The values can be reused for the creation of a new pad by selecting the grade limit object, and then selecting **Create Pad from Grade Limits** again.



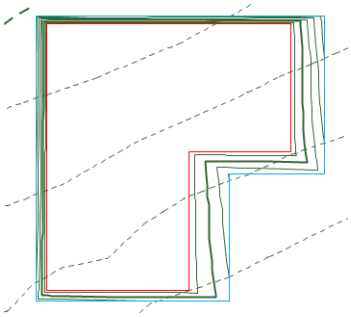
Creating a Pad
Creating a Pad with a Retaining Edge
Updating a Site Model

A L Creating a Spoil Pile Area

A spoil pile refers to an area where excess earth is used to help balance the cut and fill, so that the site does not require earth to be moved in or out. A spoil pile area applies an even thickness of cut or fill for the site. To create a spoil pile, either use the **Site Modifiers** tool, or draw a polyline and then select the **Create Objects from Shapes** command (see "Creating Objects from Shapes" on page 277).

To create a spoil pile area:

1. Create the site model and add any modifiers, such as pads and roads, and update the proposed site model by clicking **Update** from the Object Info palette of the selected site model.



2. The initial cut and fill volumes are displayed in the Object Info palette.
3. Click the **Site Modifiers** tool from the Site Planning tool set.
4. Draw the spoil pile area.

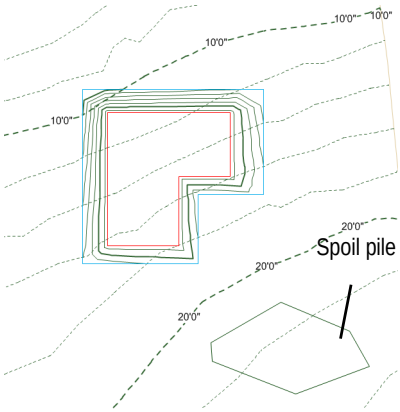
   If this is the first time the **Site Modifiers** tool has been used, the Object Preferences dialog box opens. Accept the defaults and click **OK**.
5. In the Object Info palette, select **Spoil Pile** from the **Config** field. Adjust the elevation of the spoil pile up or down depending on whether fill is required or excess fill is present.

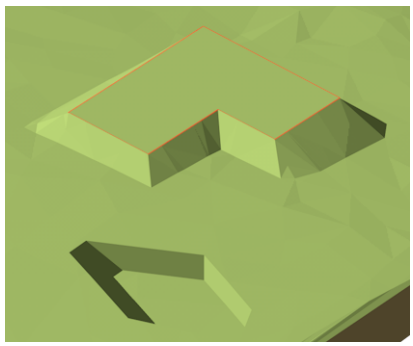
Click to show/hide the parameters.

Parameter	Description
Config	Indicates the type of modifier object

Parameter	Description
Elevation	Sets the reference elevation of the spoil pile; a positive elevation is reported as fill volume, while a negative elevation is reported as cut volume, when calculating the underlying existing or proposed site model
Apply To	Specifies whether the site modifier applies to the existing or proposed site model
Show 3D	In 3D views, represents the spoil pile with a 3D polygon; deselect this option to hide this geometry
Update Calculations	Updates the area and volume calculations displayed in the Object Info palette for the area of the site model located under the site modifier
Site model area and volume data	Displays the area and volume information for the site model area located under the site modifier; select the units for the area and volume.

6. Select the site model and click **Update** from the Object Info palette.





7. With the site model still selected, click **Update Cut and Fill Calculations** from the Object Info palette.

   Evaluate the results of the spoil pile by checking the **Net C&F Volume** results in the Object Info palette of the selected site model. If the spoil pile elevation needs adjustment, select the spoil pile and enter a new **Elevation** value in the Object Info palette. Then select the site model, click **Update**, and then click **Update Cut and Fill Calculations**.

8. Continue adjusting the spoil pile elevation until a balanced cut and fill volume is achieved.

   Display the 3D cut and fill volumes by selecting Cut and Fill as the **3D Display** in the Object Info palette of a selected site model. The cut and fill colors are specified on the Site Model tab of the Graphic Properties dialog box. In addition, a 2D polygonal representation of the cut and fill area can be displayed by selecting **2D Cut & Fill Area** from the Site Analysis tab.

Site Model Properties
Setting Site Model Graphic Properties
Inserting Stake Objects
Updating a Site Model

Creating a Texture Bed

The texture bed site modifier converts a 2D polygon into an area of the drawing that can be textured. For example, texture a lawn with grass, a terrace with bricks, or a sand trap with sand for a photorealistic rendered appearance. To

draw a texture bed, either use the **Site Modifiers** tool, or draw a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

The Renderworks product is required for creating and rendering texture beds.

 To create a texture bed:

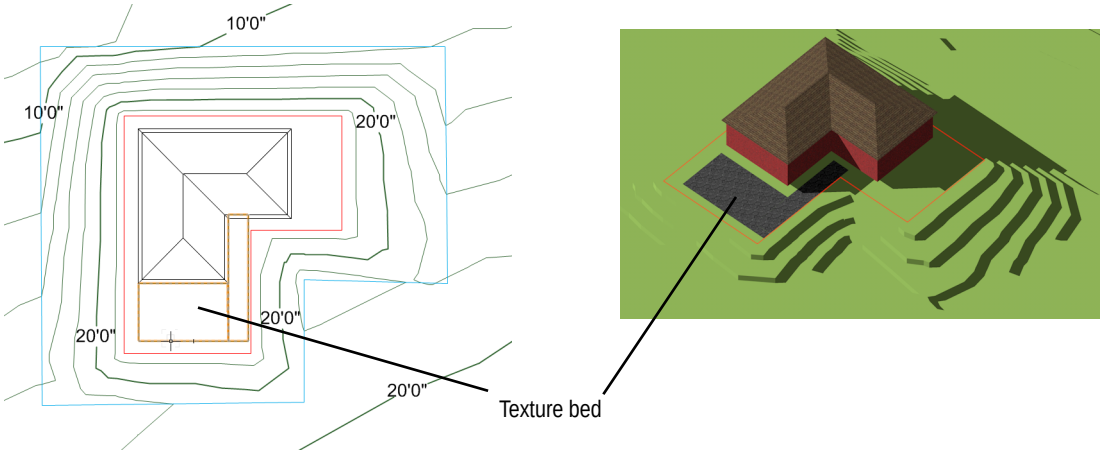
1. Click the **Site Modifiers** tool from the Site Planning tool set.
2. Draw the texture bed.
3. In the Object Info palette, select **Texture Bed** from the **Config** field.

Click to show/hide the parameters.

Parameter	Description
Config	Indicates the type of modifier object
Elevation	Sets the reference elevation of the texture bed above the site model surface; normally, set this value to zero. The texture is designed to lie on the surface of the site model when the texture bed is set to an elevation of zero.
Apply To	Specifies whether the site modifier applies to the existing or proposed site model
Texture from Class	The classes present in the drawing are listed; alternatively, create a new class by clicking New, or select the class named <Site Modifiers Class> to derive the texture and its associated surface hatch from the class of the site modifier. The indicated class should have a texture applied on the Other tab, with **Use Textures/Surface Hatches At Creation** selected. See “Applying Object Textures by Class” on page 1542.
Show 3D	In 3D views, enabling this option represents the texture bed with a 3D polygon; deselect this option to hide the geometry and display the texture bed only with its texture
Update Calculations	Updates the area and volume calculations displayed in the Object Info palette for the area of the site model located under the site modifier
Site model area and volume data	Displays the area and volume information for the site model area located under the site modifier; select the units for the area and volume. For more information, see “Site Model Properties” on page 699.

See “Textures and Shaders” on page 1505 for more information on textures.

4. The site model requires updating to reflect the texture bed modification. Select the site model and click **Update** from the Object Info palette.
5. Render the model with a Renderworks render mode to see the texture bed.



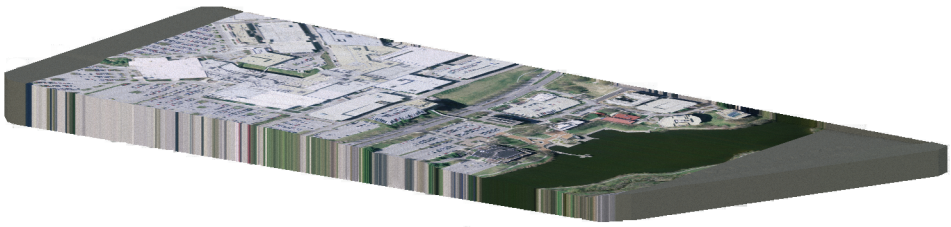
Mapping Images on the Site Model

Images can be mapped to the surface of the site model; for example, use an aerial image of the site. Unlike the texture bed, which is a site modifier, mapping of an image texture occurs from the Render tab of the Object Info palette, and is adjusted with the **Attribute Mapping** tool.

The Renderworks product is required for mapping images.

To map images on the site model:

1. Create an image texture as described in “Creating Image-based Shaders” on page 1508.
2. Select the site model. From the Object Info palette, select the image texture from the **Texture** list on the Render tab.
3. Switch to a 3D view from the **Views > Standard Views** menu, and render the site model to view the mapped image.
4. Click the **Attribute Mapping** tool from the Visualization tool set to position, move, resize, and rotate the image texture. Mapping adjustments are constrained to the 2D texture space plane; the image cannot be skewed or tilted.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Textures and Shaders
Applying a Texture to an Object
Direct Texture Mapping
Rendering with Vectorworks
Rendering with Renderworks
Updating a Site Model

AL Creating Custom Site Modifiers

As the **Site Modifiers** tool creates a pad, grade limits, spoil pile, or texture bed site modifier, the site modifier contains geometry that is placed in the Site-DTM-Modifier class. It is also possible to create site modifiers with polygons and NURBS curves; placing them in the Site-DTM-Modifier class causes them to modify the proposed site model.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

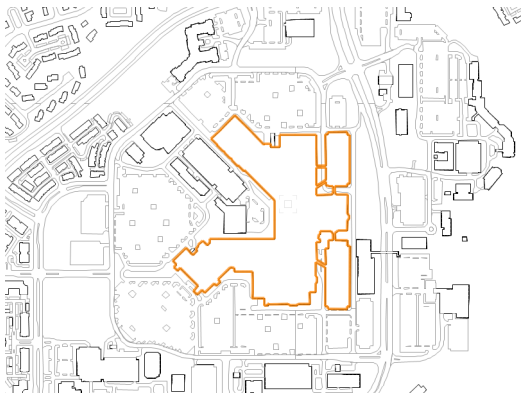
The type of site modifier created depends on the object.

- A 2D polygon acts like grade limits;
- A 3D polygon or NURBS curve acts like a pad.

  To obtain uniform sloping sides on an open 3D polygon used as a swale or break line, place the 3D polygon in the Site-DTM-Modifier class, and select **Create Grade Limits from Pad** (Vectorworks Landmark required). See "Creating Grade Limits Automatically" on page 711.

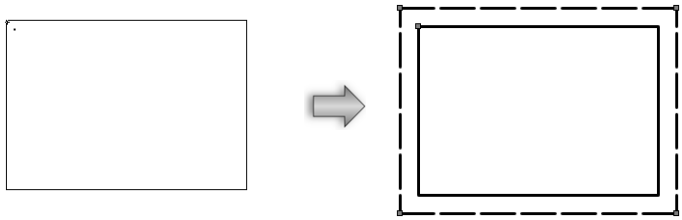
A L Creating a Massing Model

The **Massing Model** tool creates a building shell for illustrative purposes. This is an easy way to create a representative or context building when a detailed building is not required. Building models can also be created by drawing a polyline and then selecting the **Create Objects from Shapes** command (see "Creating Objects from Shapes" on page 277). A massing model can be a site modifier.



To create a massing model:

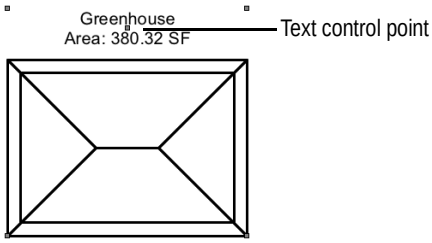
1. Click the **Massing Model** tool from the Site Planning tool set.
2. Click to begin drawing the outline of the building. Continue clicking to draw the shell polyline. Double-click, or click once at the start point, to end the polyline.



3. The massing model parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Name	Enter a building name for labeling
Show in Plan	Select whether to show the building roof or footprint in Plan view

Parameter	Description
Label	Select a text label, if any, for the building shell in Plan view; move the text control point to reposition the text 
Height	Specifies the building height (to the eave)
Floor Count	Specifies the number of floors in the building
Floor Usage Data	Opens the Floor Usage Data dialog box, for reporting the usage of massing model floors. This is useful in worksheets, for example, when reporting the usage data of the floors using the '`FloorUsageData`' parameter (see "Worksheet Functions" on page 1344). The number of floors listed depends on the **Floor Count** (if the **Floor Count** is reduced later, floor usage data will be lost). By default, each floor's usage is "Unspecified." To specify the floor usage data, select each floor and enter the **Usage Data** for that floor. A semi-colon cannot be used in the usage data, because this is the delimiter for the `FloorUsageData` function.
Wall Class	Select a class to use for wall style attributes
Use Custom Roof from Profile	Select to create a custom roof. The first roof created by the object is used as the default custom roof; if the number of stories has changed, the height of the custom roof needs to be manually adjusted.
Roof Overhang	Specifies the distance the roof extends from the building
Roof Thickness	Indicates the roof thickness
Roof Class	Select a class to use for hipped roof style attributes
Pitched Roof	Select to add a roof to the building shell
Eave Style	Select an eave style from the list
Roof Slope	Specifies the roof pitch angle in degrees
Draw Floors	Select to draw floors in the building
Slab Thickness	Specifies the floor thickness
Display Shadow in Plan View	Select to display a shadow based on the roof outline in Top/Plan view. Document-wide settings from the Plan Shadows pane on the Document Preferences dialog box are applied. Click **Shadow Settings** to open the Document Preferences dialog box to review and edit the settings (see "Plan Shadows Preferences" on page 63).
Use Site Modifiers	Adds a pad and grade limits to the base of the building and allows the building to modify the site model
Use Grade Limits	Uses grade limits to define the site modifier modification area
Grade Limits Off.	Sets the offset of the grade limits from the massing model
Gross Area	Displays the gross area value for all the floors

Parameter	Description
Vertex parameters	Edits the object vertices. See “Editing Vertex-Based Objects” on page 1002.

4. If the massing model is used as a site modifier, the proposed site model requires updating. Select the site model and click **Update** from the Object Info palette.

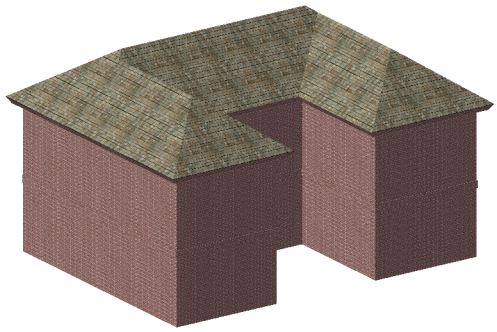
Creating a Custom Massing Model Roof
Updating a Site Model

Creating a Custom Massing Model Roof

If the standard flat or pitched massing model roof is not sufficient for portraying a more specific type of model, the roof geometry can be customized.

To create a custom massing model roof:

1. Create the massing model as described in “Creating Landscape Walls” on page 737.



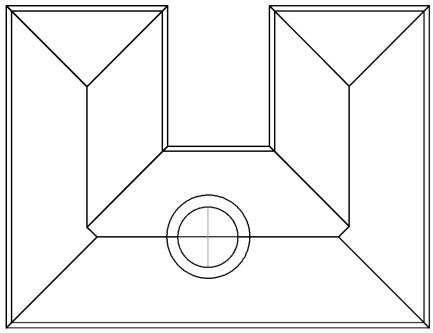
2. Select **Use Custom Roof from Profile**.
3. Select **Modify > Edit Massing Model** to enter roof profile editing mode.

   In this mode, the roof can be selected and modified by standard editing methods used in the Vectorworks program; see “Editing Objects” on page 997. The **Reshape** tool can also be used.

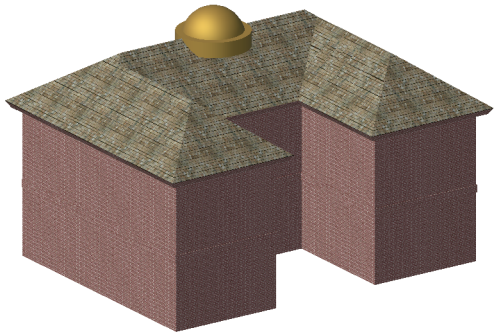
   The first roof created by the object is used as the default custom roof; if the number of stories has changed, the height of the custom roof needs to be manually adjusted. A pitched roof is treated as a roof object; edit its parameters as described in “Editing Roof Objects” on page 573. Eaves can be changed to gables, for example.

   The geometry of a pitched roof can be further edited by selecting **Modify > Edit Group** and creating cut-outs; see “Creating Cutouts in a Roof Object” on page 575. Geometry can be added to the roof (a dome or cupola), or subtracted from the roof using 3D tools. Click the **Exit Group** button (or select **Modify > Exit Group**) to return to the roof profile editing mode.

   Deleting the existing custom roof causes a copy of the current standard roof, pitched or flat, to be placed in the group when the group is exited.



4. When the roof edits have been made, click the **Exit Profile** button (or select **Modify > Exit Profile**) to return to the drawing.



Changes to the massing model may require manual updates to the custom roof; select the massing model and select **Modify > Edit Massing Model** to make any changes.

Creating a Massing Model
Editing a Group
Editing Roof Objects

A L Creating Roads

The Vectorworks Architect and Landmark products offer a variety of different tools and commands for creating roads, depending upon the complexity of the roadway. A road object can be set to be a site model modifier, and changes the proposed site model when the site model is updated.

- The **Roadway (Poly)** tool is the most straightforward, yet flexible way of creating a roadway in the Vectorworks Architect and Landmark products.
- The **Roadway (Custom Curb)** tool joins roadways for Vectorworks Architect and Landmark users. This tool can model exits, merges, traffic circles, intersections, cul-de-sacs, dead ends, and more.
- The **Roadway (NURBS)** tool offers an alternative way of inserting a road on a site model, and is available in both the Vectorworks Architect and Landmark products.
- The Vectorworks Landmark product includes commands that create a roadway from a polyline in several steps, with complete control over each step.
- For simple roadway representations in the Vectorworks Architect and Landmark products, build sections of a road with the **Roadway (Straight)** and **Roadway (Curved)** tools, and connect them with the **Roadway (Tee)** tool.

Creating Polyline Roadways

Joining Roadways with a Custom Curb
Creating NURBS Roadways
Creating a Roadway with the Vectorworks Landmark Commands
Creating and Joining Straight and Curved Roadways

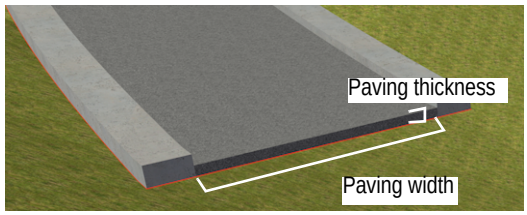
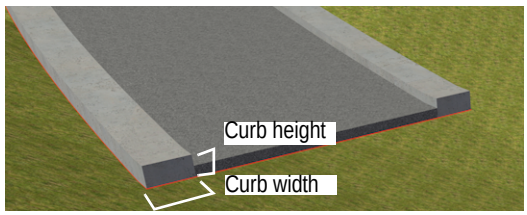
Creating Polyline Roadways

The **Roadway (Poly)** tool available creates a road defined by a path polyline or polygon. In Top/Plan view, the curbs and pavement are represented by filled polygons. In 3D views, extrudes are generated to represent the curbs and pavement. Polyline roadways can also be created by drawing a polyline or polygon and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

To create a road with the **Roadway (Poly)** object:

1. Click the **Roadway (Poly)** tool from the Site Planning tool set.
2. Click to begin drawing the roadway polyline path; click to set each polyline vertex. Double-click the mouse to create an open path polyline, or click at the start point (or press K) to complete a closed polyline. For more information on polylines, see “Creating Polylines” on page 298.
3. If this is the first time a polyline roadway is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all polyline roadways placed subsequently in this drawing. Click **OK**.
4. The parameters of the Roadway (Poly) can be edited from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Paving Width/Thickness	Sets the width and thickness of the roadway paving 
Curb Width/Height	Sets the width and height of the roadway curb 
Show Joints	Displays the roadway joints (the hidden sides where there is no curb) as dashed lines
Show Stations	Stations report the road elevation at intervals along the roadway. Select the option to show station information along the roadway curves at corner vertices; in Top/Plan view, the display includes the segment index (the number of the station, counted from the start of the roadway polyline) and the station offset from the beginning of the road curve. In 3D views, a locus displays.

Parameter	Description
Create Pad for Site Model	Creates an embedded pad site modifier around the road, following the road curvature and the elevation defined at each station point. A path polyline with many vertices will have less segmentation when the pad is created; increase the number of vertices with the **Reshape** tool if necessary to reduce the segmentation. The 3D resolution selected in the Vectorworks preferences determines the resolution of curved segments of the roadway. A site model must exist under the roadway object to create a pad modifier.
Create Grade Limits for Site Model	Creates an embedded grade limits site modifier around the road. A site model must exist under the roadway object to create a grade limits modifier.
Grade Limits Method	When creating grade limits around the road, select whether to specify the grade by offset or by slope
Grade Limits Off	For offset grade limits, specifies the offset distance of the grade limits area on all sides of the road
Grade Limits Slope	For slope grade limits, specifies the slope as an angle in document units, as a rise:run ratio (using : as the delimiter), or as a percentage, using the % symbol as a suffix. The grade limits object is created on all sides of the road so that the slope of the underlying site model is equal to the specified value.
Paving/Curb/Stations Class	Select a class for each category of roadway geometry, to control the appearance and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name (roadway items are placed in the Site-Paving class category), or select the class named <Roadway (Poly) Class>, which places the paving item in the same class as the roadway object.
Set Stations	Opens the Redistribute Stations dialog box, for creating stations along the road at a specific spacing value. Enter the spacing distance and click **OK**. Any previous station points are cleared, and new stations are created. The station spacing value can be used in worksheet reports. If the roadway is reshaped, the station points are redistributed, and elevations are adjusted accordingly.
Clear Stations	Removes all station points from the selected roadway
Send Stations to Surface	Opens the Send Stations to Surface dialog box, for creating stations along the road at a specific spacing value, while also setting the elevation of the station points to the surface of the site model. Any previous station points are cleared, and new stations are created. The station spacing value can be used in worksheet reports. If the roadway is reshaped, the station points are redistributed, and elevations are adjusted accordingly.
Align Stations Vertically	Opens the Align Stations Vertically dialog box, which displays the grade between specified station points. Enter the station point starting number and ending number (Station 1 is the station at the beginning of the roadway). The selected stations are highlighted on the drawing. Click **OK**. The stations between the starting and ending stations are fit to the slope grade.
Vertex parameters	Edits the vertices of the path object that the roadway is based upon; see “Editing Vertex-Based Objects” on page 1002. The elevation of station points can also be viewed and edited by selecting Stations from the **Move** list.

Parameter	Description
Roadway parameters	Displays roadway parameters for the selected road, such as area and length

5. If necessary, use the **Reshape** tool to modify the locations of the vertices after object creation, or use the vertex editing controls on the Object Info palette to move the vertices, or change the elevation of station points.
6. Update the site model. Select the site model and click **Update** from the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Updating a Site Model
Creating Roads

AL Joining Roadways with a Custom Curb

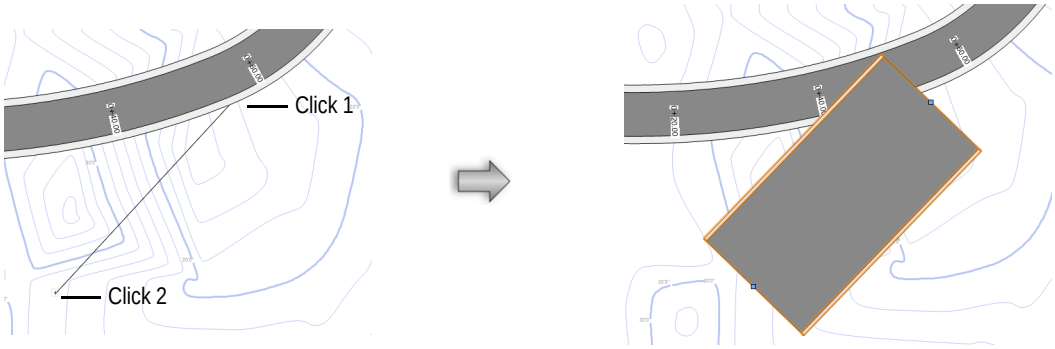
The **Roadway (Custom Curb)** tool creates sections of roadways that join other roadways in various ways, such as intersections, exits, merges, traffic circles, intersections, cul-de-sacs, dead ends, and more.

The tool creates a free-form roadway shape that is extremely flexible, and can be reshaped to fit the needs at the particular location. The curb can be hidden along the sides of the roadway object, where it connects to other roadways.

Custom curb roadways can also be created by drawing a polyline or polygon and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

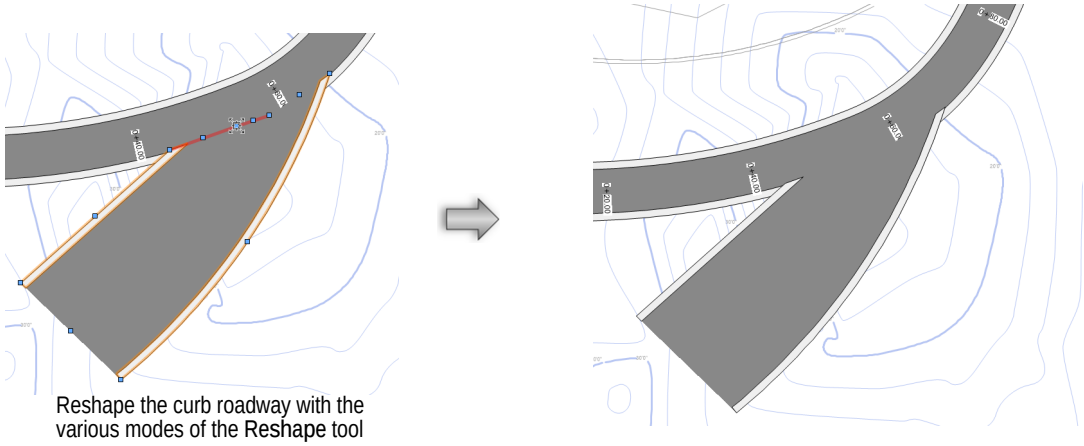
To create a custom curb roadway:

1. Click the **Roadway (Custom Curb)** tool from the Site Planning tool set.
2. Click to set the starting point of the roadway. Click again to set the ending point. These clicks determine the initial length and orientation of the curb roadway. When selected, the roadway displays with two control points which define the length and orientation.



3. Use the **Reshape** tool to reshape the curb roadway as described in “Reshaping Objects” on page 1043. Roadway vertices can be repositioned, added, and deleted; however, the two control points that define the roadway cannot be reshaped. The **Change Vertex** mode of the **Reshape** tool can switch vertices to arc vertices to use curves in the curb roadway. The **Hide or Show Edges** mode of the **Reshape** tool can hide and show the portions of the curb where the roadway intersects with other roadway objects.

   The curb roadway can be reshaped while in path editing mode. Select **Modify > Edit Roadway (Custom Curb)**, or right-click (Windows) or Ctrl-click (Mac) on the curb roadway and select **Edit Path** from the context menu.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Reshape the curb roadway with the various modes of the **Reshape** tool

4. The Roadway (Custom Curb) parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Road Length	For worksheet reporting purposes, specifies the length of the roadway; however, this setting has no effect on the actual road length, which is edited by adjusting the control points with the **Selection** tool
Paving Width	For worksheet reporting purposes, specifies the width of the roadway; however, this setting has no effect on the actual road width, which is edited by adjusting vertices with the **Reshape** tool
Curb Height/Width	Indicates the width and height of the roadway curb
Paving Thickness	Sets the thickness of the road paving
Rise	Sets the difference in height between the start and end of the roadway segment
Show Joints	Displays the roadway joints (the hidden sides where there is no curb) as dashed lines
Use Site Modifiers	Creates a pad and grade limits to modify the site model
Create Grade Limits for Site Model	Uses grade limits to define the site modifier modification area
Left/Right Grade Limits Off	Controls the grade limit offset distance on either side of the road
Paving/Curb Class	Select a class for each category of roadway geometry, to control the appearance and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name (roadway items are placed in the Site-Paving class category), or select the Roadway (Custom Curb) class, which places the paving item in the same class as the curb roadway object.

5. If the roadway sections include pads or grade limits, and a site model exists, select the site model and click **Update** from the Object Info palette.

Updating a Site Model
Creating Roads

Creating NURBS Roadways

The **Roadway (NURBS)** tool uses NURBS curves to define the road shape. However, the Roadway (NURBS) object does not represent accurate alignments of low-speed roadways, which are typically constructed with arcs (constant radius) and tangents (straight lines); use the **Roadway (Poly)** tool for this.

NURBS roadways can also be created by drawing a polyline or polygon and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

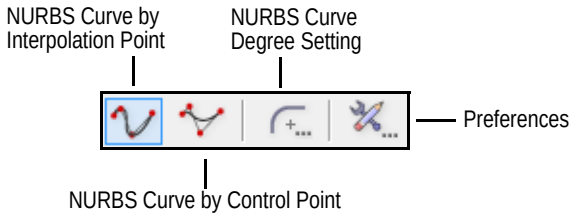
Here is a suggested process for creating a NURBS roadway:

- Create the roadway.
- Select the **Send to Surface** command to set the road to the existing terrain.
- Select **Redistribute and Smooth Curve** from the Object Info palette of the selected roadway once or twice to smooth out the road slightly, so it does not follow every hill and valley in the existing terrain.
- Manually reshape the road in 3D as required (to match other road elevations, garage elevations, and so on).
- Only select **Redistribute Stations** for more or fewer station annotations. Fewer stations are easier to work with when sketching and reshaping, but a road construction company might require more stations to be specified. **Redistribute Stations** is also useful to ensure that the station spacing is equal after reshaping the road.

 To create a road with the **Roadway (NURBS)** object:

1. Click the **Roadway (NURBS)** tool from the Site Planning tool set.

   The following modes are available.



   See “Creating NURBS Curves” on page 328.

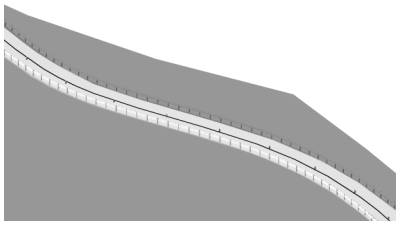
2. Click in the drawing file to set the start point of the road, and then click to set the position of each additional NURBS vertex. Double-click the last point to complete creation of the roadway (NURBS) object.
3. If this is the first time a NURBS roadway is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all NURBS roadways placed subsequently in this drawing. Click **OK**.
4. The Roadway (NURBS) properties can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Z height	Adjusts the elevation of the entire road
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file’s content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See “Using Text Styles” on page 393 and “Formatting Text” on page 389.
Road Length	Displays the length of the road

Parameter	Description
Paving Width	Specifies the overall width of the paved road surface (without curbs)
Paving Height	Specifies the road paving thickness (without curbs)
Curb Width	Specifies the curb or shoulder width
Curb Height	Specifies the curb or shoulder height
Left Guardrail	Adds a guardrail to the left side of the road
Right Guardrail	Adds a guardrail to the right side of the road
Show Stations	The NURBS curve vertices are used as the station points; select to display the station points. In 2D, stations display as loci with text displaying the point elevation. In 3D, the stations display as round stakes.
Create Pad for Site Model	Creates a pad modifier for the Roadway (NURBS) object
Create Grade Limits for Site Model	Creates a grade limits modifier for the Roadway (NURBS) object, to control the zone where the terrain transitions between the existing elevation and the road elevation
Grade Limits Method	Specify whether to create offset or sloped grade limits
Grade Limits Off.	When creating offset grade limits, specify the distance from the road to the grade limits
Grade Limits Slope	When creating sloped grade limits, specify the batter slope
Paving/Curb Class	Select a class for each category of roadway geometry, to control the appearance and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name (roadway items are placed in the Site-Paving class category), or select the Roadway (NURBS) class, which places the paving item in the same class as the NURBS roadway object.
Redistribute Stations	Redistributes the stations at a specified spacing interval
Redistribute and Smooth Curve	Redistributes the station points at a specified spacing interval and smooths the NURBS curve
Send Stations to Surface	Click to set the Z value of the stations to the surface of the site model. The stations can be set to either the surface of the existing site model, or to the surface of the proposed site model, which may have been altered by modifiers. If the stations are sent to the surface of a proposed site model which has been changed by modifiers, it may be necessary to deselect **Create Pad for Site Model** and **Create Grade Limits for Site Model** for the Roadway (NURBS) in order to prevent site modifier errors caused by crossing pads or grade limits.
Align Stations Vertically	Sets the elevation of the indicated station points to a specific plane. As the starting and ending stations are indicated, a red line previews the included station points. The **% Grade** indicates the grade that will be set between station points.
NURBS curve parameters	Edits the roadway NURBS curve

5. If necessary, use the **Reshape** tool to modify the locations of the vertices after object creation, or use the vertex editing controls on the Object Info palette to move the vertices or change the degree of vertices.
6. Update the site model. Select the site model and click **Update** from the Object Info palette.



Updating a Site Model
Creating Roads

Creating a Roadway with the Vectorworks Landmark Commands

The Vectorworks Landmark road commands create a roadway with a multi-step procedure. The centerline of the road is defined with a polyline, and then stakes are added. The elevation of the stakes, which are initially set to the surface of the site model, can be modified before the road is created. Finally, the site model is updated and modified by the road object.

Creating the Road Centerline
Placing Station Points Along the Road Polyline
Setting Stake Elevations
Creating the Road

Creating the Road Centerline

The road design elements can be created on a separate layer to facilitate modifications later.

To create the road centerline:

1. If desired, create a new layer for the road by selecting **Tools > Organization**. The layer scale should be the same as the site model layer scale. Set the site model layer to visible, and select **View > Layer Options > Show/Snap Others**.
2. Using the **Polyline** tool, create a polyline that represents the road shape.

   For more information on polylines, see “Creating Polylines” on page 298.

   Interior vertices of the polyline must be arc-smoothed (no Bézier or cubic vertices).

   Smooth vertices with the **Modify > Poly Smoothing > Arc Smoothing** command.

The road must be contained within the site model grade limits.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Placing Station Points Along the Road Polyline
Setting Stake Elevations
Creating the Road

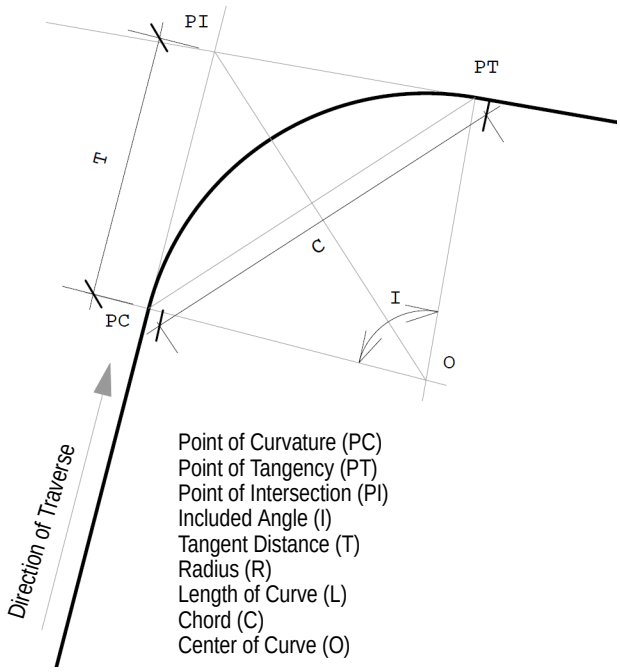
Placing Station Points Along the Road Polyline

The Vectorworks Landmark product places station points (stake objects) along the selected polyline at a specified interval.

To place station points:

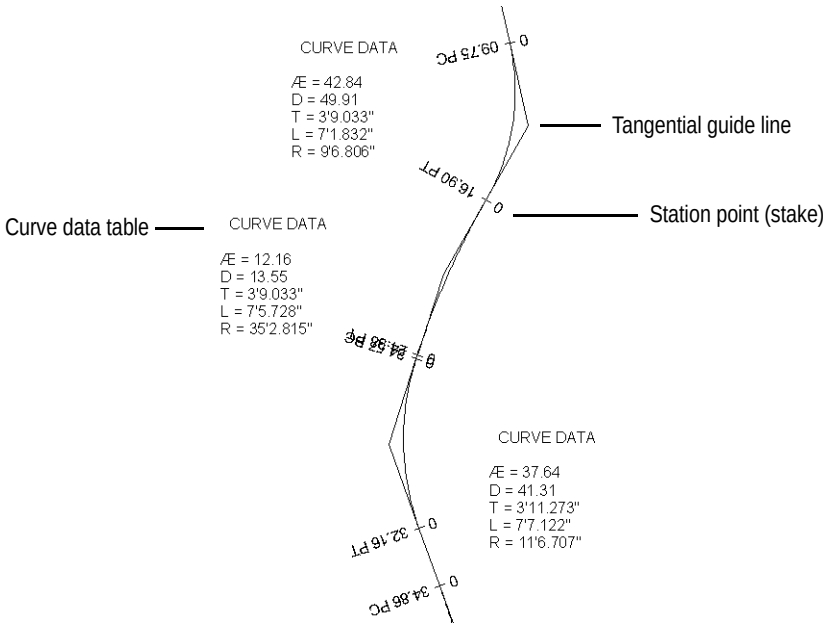
1. Select the polyline, and then select the **Station on Polyline** command from the appropriate menu:
   - Designer workspace: **AEC > Roads > Station on Polyline**.
   - Landmark workspace: **Landmark > Roads > Station on Polyline**
2. The Place Station Points Along Polyline dialog box opens. This command places stations at 100’ (or 30.48 m) intervals. In addition, partial station points can be placed at smaller intervals. Enter the interval between partial stations and select a labeling option for the curve information.

Click to show/hide the parameters.

Parameter	Description
Partial station interval	Indicates the spacing interval between partial station points
Create individual curve labels	Creates curve data tables at each polyline curve. The curve labels include the following data: 
Create circular curve data table	Creates a Station Worksheet with the curve data; the worksheet number corresponds to the polyline number
No Curve Data	No curve data tables or tangent guide lines are created
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Click **OK** to create the station points along the polyline. If this is the first time a stake has been placed in the file, the Object Properties dialog box opens. Accept the default values and click **OK**. Stake properties are described in “Inserting Stake Objects” on page 769.

   Each station point is labeled. If curve data tables were created, they can be selected and moved if necessary.



4. The stakes have all been set to the elevation of the site model; the next step is to automatically set the road stake elevations as described in “Setting Stake Elevations” on page 730.

Inserting Stake Objects
Creating the Road Centerline
Setting Stake Elevations
Creating the Road

Setting Stake Elevations

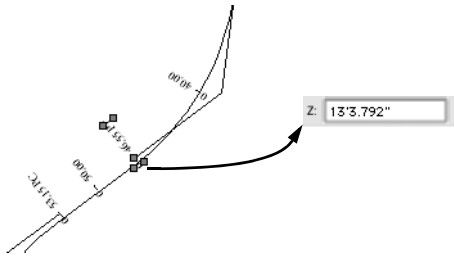
In “Placing Station Points Along the Road Polyline” on page 729, the stake elevations were set according to the site model; in real situations, a road is often set to a grade other than natural grade. The **Align Stakes Vertically** command sets the elevation of the stakes to a specific plane.

To set the stake elevations with a linear alignment:

1. Set the elevation of only one of the terminal stakes (one of the stakes at the ends of the segment to align) by selecting the stake and entering its elevation in the Object Info palette. If a site model was present when the **Station on Polyline** command was executed, the elevation of the stake is set to the proposed site model elevation for reference.
2. In the segment to align, press the Shift key to select both the first and last stake (the stakes at either end of the desired elevation alignment), and then select the **Align Stakes Vertically** command from the appropriate menu:
   - Designer workspace: **AEC > Roads > Align Stakes Vertically**
   - Landmark workspace: **Landmark > Roads > Align Stakes Vertically**

   The Align Stakes Vertically dialog box opens.
3. Enter either a slope value as a rise over run ratio for the road (using a forward slash (/) as a delimiter), or the elevation of the last stake, and click **OK**.

The elevations of the stakes located between the selected stakes are set to the specified plane.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating the Road Centerline
Placing Station Points Along the Road Polyline
Inserting Stake Objects
Creating the Road

Creating the Road

This procedure creates the road by automatically making a single polyline roadway, or by creating curved or straight road objects between each stake. The road follows the centerline polyline, and is set to the elevations specified by the stakes.

To create the road:

1. Select the center polyline or any of the station points, and select the **Create Road from Stakes** command from the appropriate menu:
   - Designer workspace: **AEC > Roads Create Road from Stakes**
   - Landmark workspace: **Landmark > Roads > Create Road from Stakes**

   The Create Road from Stakes dialog box opens. Enter the road parameters.

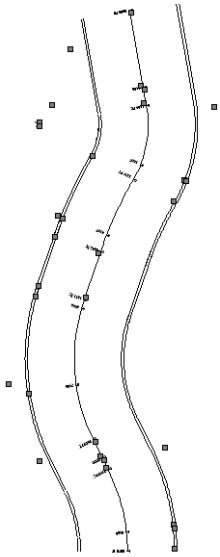
Click to show/hide the parameters.

Parameter	Description
Select the result type	
A single Roadway (Poly) object	Creates the road as a Roadway (Poly) object
A set of Roadway (Curve) and Roadway (Straight) objects	Creates the road as a series of curved and straight roadway objects along the stations
Width	Specifies the road width between the curbs
Curb Height/Width	Specifies the height and width of the road curbs
Paving Thickness	Sets the thickness of the road paving

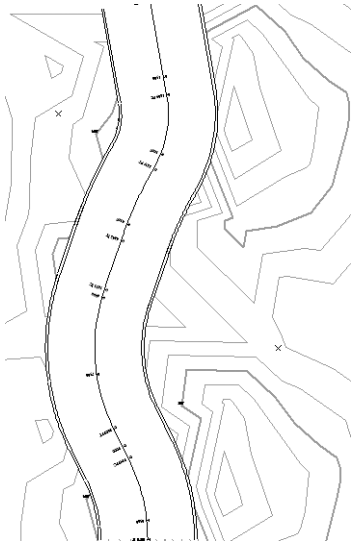
2. Click **OK**. The polyline and stakes are used to create a Roadway (Poly) object, or a series of road section between adjacent stakes. If this is the first time a road (or a straight or curved road) has been created in the file during this session, the Object Properties dialog box opens. Accept the default values and click **OK** in both dialog boxes.
3. If a single roadway was created, create a pad and grade limits for the roadway to modify the site model by selecting the options on the Object Info palette. See “Creating Polyline Roadways” on page 722.

   If a set of straight and curved roads was created, the straight and curved sections of the road need to become site modifiers by selecting the road sections and then selecting **Use Site Modifiers** in the Object Info palette. Use the **Custom Selection** command to select the straight and curved road sections; see “Creating Custom Selection Scripts” on page 1771. The grade limits offset can be set manually, or the grade limits can be turned off and
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

created separately using the **Create Grade Limits from Pad** command (see "Creating Grade Limits Automatically" on page 711). When site modifiers are used, each road section contains a pad (see "Creating a Pad" on page 706 and "Creating Grade Limits" on page 710).



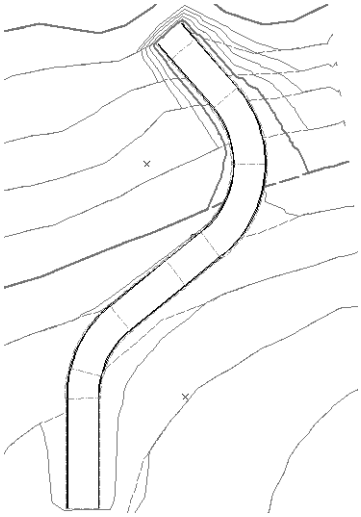
Stakes should not be moved (except in elevation) when using the **Create Road from Stakes** command. If an additional stake is required, add a vertex point to the polyline, provide an elevation value, and run the **Create Road from Stakes** command again.

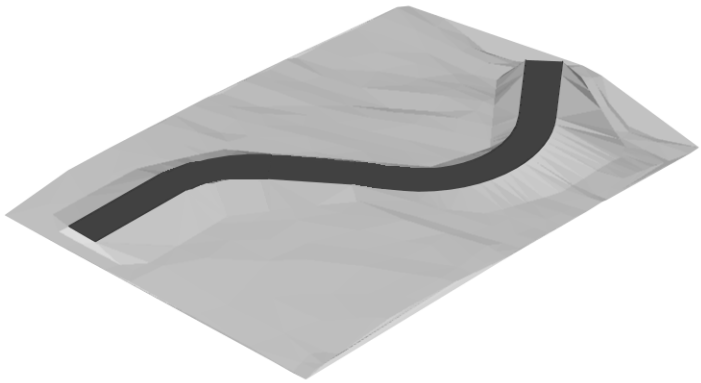


4. The proposed site model requires updating. Select the site model and click **Update** from the Object Info palette.

   If the road was created on a different layer from the site model, click Site Model Settings from the Object Info palette and ensure that **Use Site Modifiers on All Layers** is selected.

   The proposed site model is updated. If **Use Site Modifiers** was selected for the road sections, the contour lines are adjusted to rise or fall to meet the level of the road.





Use class settings to add a solid fill color to the road

Updating a Site Model
Creating Roads

A L Creating and Joining Straight and Curved Roadways

Simple roadway tools allow you to assemble and join straight and curved roadway sections. The roadway tools can modify the site model.

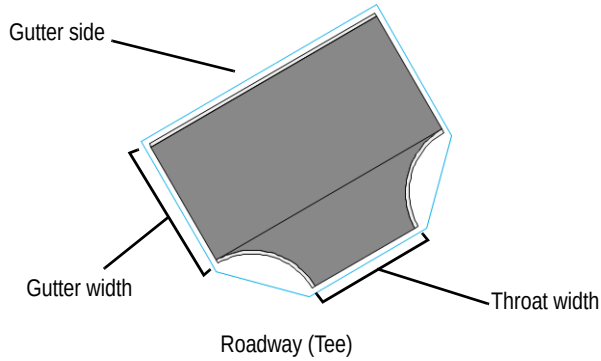
 To create a straight roadway section:

 To create a curved roadway section:

 To create a roadway intersection:

1. Click the **Roadway (Straight)** tool, **Roadway (Curved)** tool, or **Roadway (Tee)** tool from the Site Planning tool set.

   The desired tool depends on the portion of roadway section to create.
2. Create the roadway section in the drawing.
   - If placing a straight roadway section, click once to define the starting point of the road, and then click again to set the ending point.
   - If placing a curved or tee roadway section, click to place the object in the drawing, and click again to set the object's rotation.
3. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
4. Continue to assemble the roadway by placing curved and straight sections, as well as tee intersections.
5. The roadway properties can be edited in the Object Info palette.



Click to show/hide the parameters.

Parameter	Description
Z	Sets the elevation of the first stake in the roadway segment
Radius (curved and tee roads)	Specifies the arc radius (measured to the center of the road) of a curved roadway segment; the locus indicates the center of the arc (or for tee roads, indicates the center of each arc)
Width (curved roads)	Sets the width of a curved roadway segment
Throat Width (tee roads)	Indicates the width of a tee “throat” section
Road Length (straight roads)	Specifies the length of a straight roadway segment
Paving Width (straight roads)	Sets the width of a straight roadway segment
Curb Height/Width	Indicates the width and height of the roadway curb
Paving Thickness	Sets the thickness of the road paving
Rise	Sets the difference in height between the start and end of the roadway segment
Show Joints	Displays the roadway joints (the hidden sides where there is no curb) as dashed lines
Use Site Modifiers	Creates a pad and grade limits to modify the site model
Left/Right Grade Limits Off. (curved and straight roads)	Controls the grade limit offsets on either side of the road
Sweep (curved and tee roads)	Sets the degree (arc angle) of the sweep of the segment
Draw Gutter Curb (tee roads)	Draws a curb on the gutter side of a tee roadway; deselect the option to hide the curb
Gutter Width (tee roads)	Specifies the width of the gutter for tee roadways
Paving/Curb Class	Select a class for each category of roadway geometry, to control the appearance and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name (roadway items are placed in the Site-Paving class category), or select the Roadway class, which places the paving item in the same class as the roadway object.

6. If the roadway sections include pads or grade limits, and a site model exists, select the site model and click **Update** from the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Updating a Site Model
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Roads

A L Creating Landscape Walls

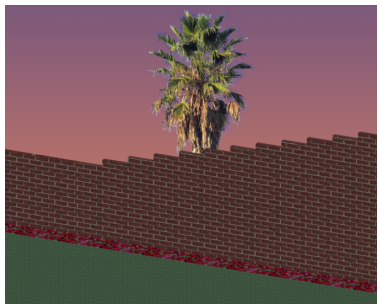
The Vectorworks Architect and Landmark products include the ability to create a stepped wall. The Vectorworks Landmark program also contains the ability to create a retaining wall site modifier, and several ways of creating landscape walls with the landscape wall tools. The stepped wall and retaining wall site modifier are based on walls created with the **Wall** or **Round Wall** tool, offering the benefits and flexibility of the architectural wall tools.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Stepped Walls
Creating Retaining Walls
Landscape Wall Tools

A L Creating Stepped Walls

Stepped walls add wall peaks to existing walls created with the **Wall** tool or **Round Wall** tool. The existing wall can be a single wall or a chain of multiple wall segments that are L-joined to each other. Stepped walls can have a constant rise at each interval, or conform to underlying terrain with a variable rise, such as free-standing landscape walls or fences.

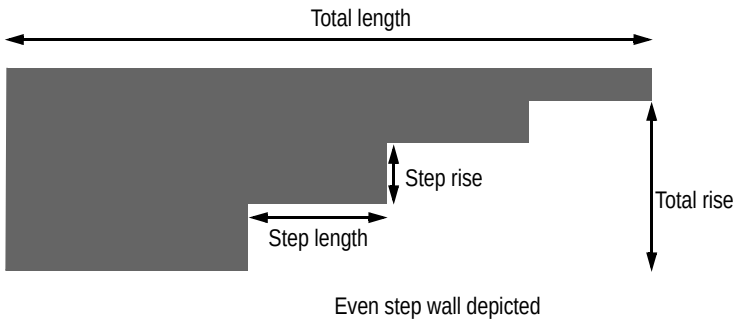


To create a stepped wall:

1. Select the wall or multiple, L-joined walls.
2. Select the **Create Stepped Wall** command from the appropriate menu:
   - Architect workspace: **AEC > Create Stepped Wall**
   - Landmark workspace: **Landmark > Architectural > Create Stepped Wall**

   The Create Stepped Wall dialog box opens. Set the stair parameters.

   The options available depend on the selected **Step Style** (even steps or terrain steps). If the parameters are set so that the steps are uneven, the top step is truncated.



Even step wall depicted

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Even Steps	Creates a stepped wall with a constant rise and wall peaks at even intervals
Preview	Previews the selected configuration
Total Length	Displays the total length of the stepped wall to be created
Number of Steps	Displays the total number of steps to be created
Calculator	The steps are created based on three parameters: step length, step rise, and total rise. Based on the entry of two of these parameters, the third is automatically calculated. Click to place the calculator at the parameter that should be automatically calculated, and enter the values for the other two parameters.
Step Length	Sets the length of one step
Step Rise	Sets the rise of one step
Total Rise	Specifies the total rise value
Apply Steps to	Steps can be applied to either the top or bottom of the wall, or both
Top of Wall	Steps apply to the top of the wall
Horizontal Offset	Specifies the horizontal offset of the top steps; positive values move the steps to the left, while negative values move them to the right. This allows the last step to be shorter or longer, for example.
Bottom of Wall	Steps apply to the bottom of the wall
Horizontal Offset	Specifies the horizontal offset of the bottom steps; positive values move the steps to the left, while negative values move them to the right.
Reverse Direction	Sets whether steps should be started from the beginning of the wall (option is deselected) or end of the wall (option is selected)
Terrain Steps	
Total Length	Displays the total length of the stepped wall to be created
Number of Steps	Displays the total number of steps to be created
Step Length	Sets the length of one step
Vertical Offset	Specifies the offset distance from the terrain surface; an offset of zero keeps the steps at the surface, while a positive value raises them above the surface
Apply Steps to	
Top of Wall	Applies steps to the top of the wall, mirroring the terrain
Horizontal Offset	Specifies the horizontal offset of the top steps; positive values move the steps to the left, while negative values move them to the right. This allows the last step to be shorter or longer, for example.
Bottom of Wall	Terrain steps are always applied to the bottom of the wall, since the steps follow the terrain
Horizontal Offset	Specifies the horizontal offset of the bottom steps; positive values move the steps to the left, while negative values move them to the right.
Place Steps Above Terrain	The steps are reversed so that they are above the terrain

3. Click **OK** to create the stepped wall.

The **Send to Surface** command can be used on walls; it sets the bottom Z value at the start of the wall to the existing terrain surface (see “Sending Objects to the Site Model Surface” on page 705).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Walls
L Wall Joins
Creating Retaining Walls
Landscape Wall Tools

Creating Retaining Walls

A retaining wall site modifier can be created based on existing walls created with the **Wall** tool or **Round Wall** tool. The existing wall can be a single wall or a chain of multiple wall segments that are L-joined to each other. The retaining wall created can modify the site model, and the terrain can be sculpted as desired around the retaining wall.

Once created, the retaining wall site modifier is not associated with the wall object used to create it.

To create a retaining wall site modifier:

1. Select the wall or multiple, L-joined walls.

   While walls are typically be used as the basis for creating a retaining wall, other valid selections include a hardscape object or an extrude object.

2. Select the **Create Retaining Wall Site Modifier** command from the appropriate menu:
   - Designer workspace: **AEC > Terrain > Create Retaining Wall Site Modifier**
   - Landmark workspace: **Landmark > Create Retaining Wall Site Modifier**

   The Create Retaining Wall Site Modifier dialog box opens. Set the initial location of the site modifiers within the retaining wall. There are three distinct areas that can modify the site model:
   - the pad at the bottom of the wall, which can be offset from the top or bottom of the wall, or simply follow the terrain
   - the left and right side edges of the wall, which can be offset from the top or bottom of the wall, or follow the terrain
   - the starting and ending edges of the wall

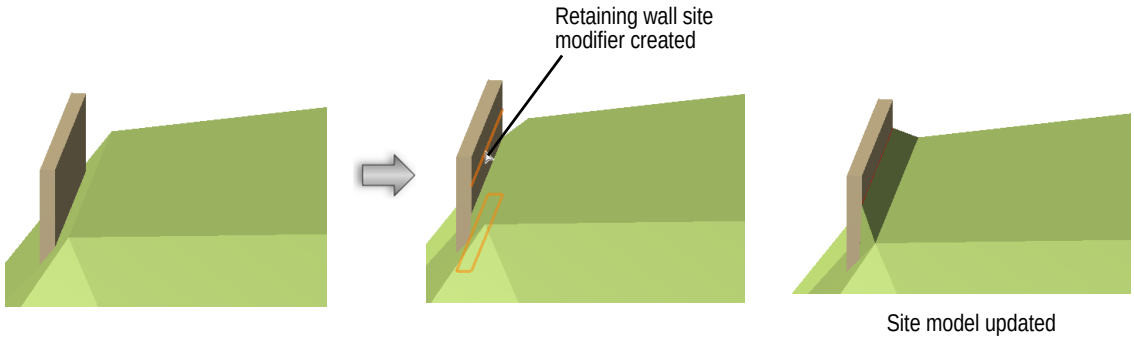
   The left and right sides of the wall are determined by the wall direction as it was drawn.

Click to show/hide the parameters.

Parameter	Description
Diagram	This wall schematic shows the location of the pad modifier at the bottom of the wall (1), the left (2) and right (3) edge modifiers, and the start (4) and end (5) edge modifiers; it also shows the wall start and end, which determines the wall direction and therefore which edge is left and right
Pad	Specifies the vertical offset and shape of the modifier pad; the modifier can be offset from the top or bottom of the wall and follow top or bottom wall peaks, or can simply follow the current terrain. The pad offset can control the cut and fill values produced by the site modifier.
Offset Below Wall Top	Sets the pad modifier to be offset below the top of the wall by the specified value
Follow Top Wall Peaks	The modifier shape follows the wall peaks at the top of the wall. If a wall has vertical peaks, the modifiers will be nearly vertical.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Offset Above Wall Bottom	Sets the pad modifier to be offset above the bottom of the wall by the specified value
Follow Bottom Wall Peaks	The modifier shape follows the wall peaks at the bottom of the wall. If a wall has vertical peaks, the modifiers will be nearly vertical.
Follow Terrain	The pad modifier follows the terrain location (offset and shape) along the bottom of the wall
Side Edges	
Left/Right Modifier Edge	Select whether to create the edge modifier objects on the left and/or right sides of the retaining wall. Edge modifiers sculpt the terrain at the left or right of the retaining wall sides according to the parameters set. The modifier can be offset from the top or bottom of the wall and follow top or bottom wall peaks, or can simply follow the current terrain.
Offset Below Wall Top	Sets the edge modifier to be offset below the top of the wall by the specified value
Follow Top Wall Peaks	The modifier shape follows the wall peaks at the top of the wall. If a wall has vertical peaks, the modifiers will be nearly vertical.
Offset Above Wall Bottom	Sets the edge modifier to be offset above the bottom of the wall by the specified value
Follow Bottom Wall Peaks	The modifier shape follows the wall peaks at the bottom of the wall. If a wall has vertical peaks, the modifiers will be nearly vertical.
Follow Terrain	The edge modifier follows the terrain location (offset and shape) along the side of the wall
Start/End Edges	The start and end edge modifiers define how the terrain wraps around the retaining wall sides
Include Start Modifier Edge	Select whether to create an edge modifier object at the starting edge of the wall
Include End Modifier Edge	Select whether to create an edge modifier object at the ending edge of the wall

3. Click **OK** to create the retaining wall site modifier. The original wall(s) remain in the drawing; the site modifier is a new, separate object.
4. Update the site model to apply the changes (select the site model and click **Update** from the Object Info palette).



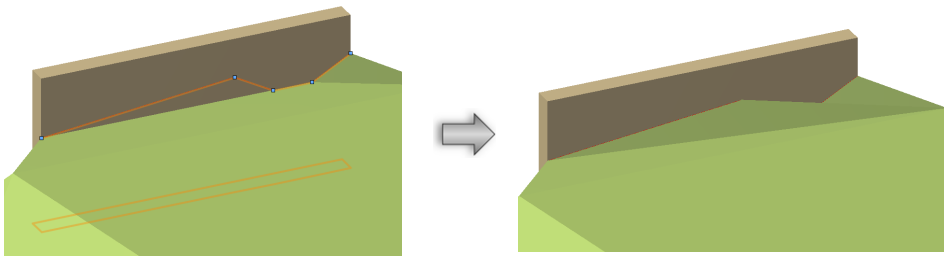
Editing Retaining Walls

Retaining wall parameters can be edited from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Apply To	Select whether the retaining wall modifies the existing or proposed site model
Width	Determines the distance between the left and right edges of the retaining wall
Pad Offset	Specifies the offset of the pad from the Z elevation of the retaining wall site modifier object itself; the offset can control the cut and fill values produced by this site modifier
Include Left/Right/Start/ End Modifier Edge	Toggles the modification of the site model by the selected modifier edge parameters
Vertex parameters	Edits the path vertices; see “Editing Vertex-Based Objects” on page 1002
Move	Select the portion of the retaining wall modifier to edit, and then edit its **Elevation**, or scroll through its vertices with the left and right arrows and change the selected vertex **Elevation**
Edit	For vertex selections made in **Move**, scrolls through the vertices, highlighting the currently selected vertex. Click the center button to highlight the selected vertex.
Elevation	Sets the elevation of the modifier item selected in **Move** or the vertex selected in **Edit**

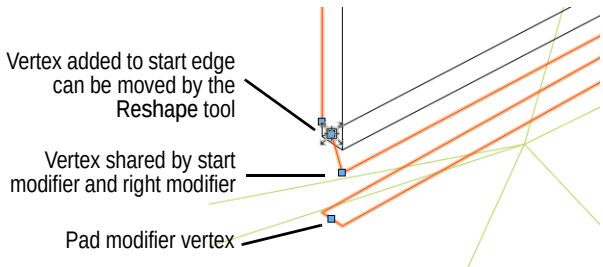
The retaining wall site modifier consists of four modifier edges, joined by common vertices. The left and right edges of the retaining wall site modifier can be reshaped with the **Reshape** tool, similar to reshaping walls (see “Reshaping Walls” on page 536). Move vertices, and add or delete vertices to reshape the retaining wall.



Reshape right edge with the Reshape tool in a 3D view

Site model updated

The start and end edges cannot be directly edited with the **Reshape** tool. However, a vertex added to the left or right edge can be edited by the **Reshape** tool (select Start/End Modifier Edge from the **Move** list in the Object Info palette, and click **Add Vertex**). Adding vertices to the start and end edges allows them to be reshaped as needed.



The **Send to Surface** command can be used to send either the retaining wall modifier edge or the pad to the surface of the site model.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Walls
Wall Direction
Reshaping Walls
Joining Walls
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Stepped Walls
Landscape Wall Tools
Reshaping Objects
Sending Objects to the Site Model Surface
Updating a Site Model

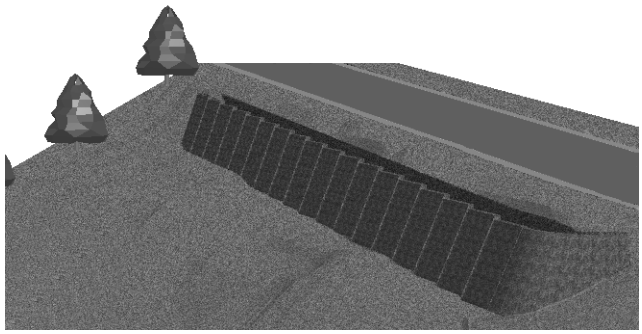
Landscape Wall Tools

The Vectorworks Landmark program includes three tools for creating landscape walls.

A landscape wall object (retaining wall) can be set to be a site model modifier. It changes the proposed site model when the site model is updated.

To draw straight, arc, and Bézier landscape walls, either use the landscape wall tools, or draw a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277). Place the different types of landscape walls next to each other to achieve a particular design or effect.

To create a row of straight and curved landscape walls, use the **Create Objects from Shapes** command.

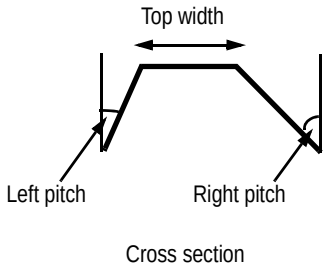


Straight Landscape Walls

To insert a straight landscape wall:

1. Click the **Landscape Wall** tool from the Site Planning tool set.
2. Click and drag to define the length and angle of the landscape wall. Click again to set the end point. If this is the first time the tool is used during this session, the Object Properties dialog box opens. Accept the default values and click **OK**.

   The straight landscape wall parameters can be edited in the Object Info palette.



Cross section

Click to show/hide the parameters.

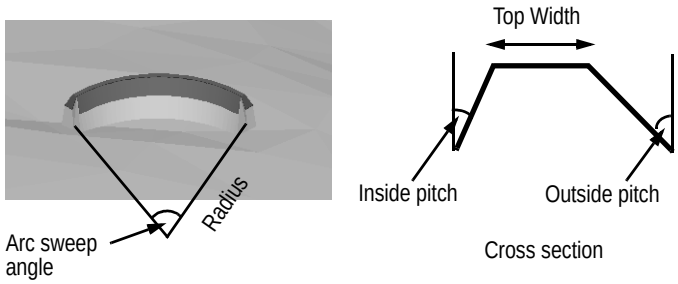
Parameter	Description
Z	Elevation of starting point of wall
Rot	Angle of rotation of wall
Length	Total length of all wall sections in the object
Top Width	Width of wall at top (see diagram)
Step Height	When the wall starting point and ending point are different, this parameter indicates the height of the step between sections
Wall Height	Wall height above Z value
Rise	Difference in height between wall starting and ending points
Left/Right Pitch	Angle of wall on left and right sides (see diagram); for a straight wall with site modifiers, the left side is always the side that retains earth
Use Site Modifiers	Adds a pad and grade limits to each wall section and allows the landscape wall to modify the site model
Use Grade Limits	Uses grade limits to define the site modifier modification area
Left/Right Grade Limits Off.	Controls the grade limit offsets on either side of the wall (applies only when site modifiers are used)
Show Joints	Displays the connecting joints between wall sections

Arc Landscape Walls

To insert an arc landscape wall:

1. Click the **Landscape Wall Arc** tool from the Site Planning tool set.
2. Click to define the placement of the arc landscape wall and click again to set the rotation. If this is the first time the tool is used during this session, the Object Properties dialog box opens. Accept the default values and click **OK**.

   The arc landscape wall parameters can be edited in the Object Info palette.



Click to show/hide the parameters.

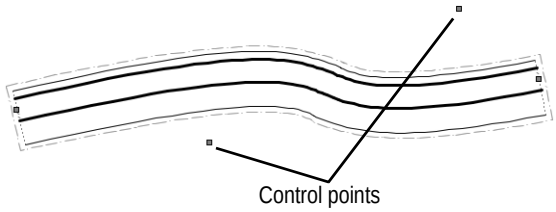
Parameter	Description
Z	Elevation of starting point of wall
Radius	Radius of all wall sections
Top Width	Width of wall at top (see diagram)

Parameter	Description
Step Height	When the wall starting point and ending point are different, this parameter indicates the height of the step between sections
Wall Height	Wall height above Z value
Rise	Difference in height between wall starting and ending points
Inside/Outside Pitch	Angle of wall on the inside and outside (see diagram); for an arc wall with site modifiers, the side of the wall that retains earth must be selected in **Retained Side**
Show Joints	Displays the connecting joints between wall sections
Retained Side	For an arc wall with site modifiers, select the side of the wall that retains earth
Arc Sweep	Angle, in degrees, of arc (see diagram)
Use Site Modifiers	Adds a pad and grade limits to each wall section and allows the landscape wall to modify the site model
Use Grade Limits	Uses grade limits to define the site modifier modification area
Inside/Outside Grade Limits Off.	Controls the grade limit offsets on either side of the wall (applies only when site modifiers are used)

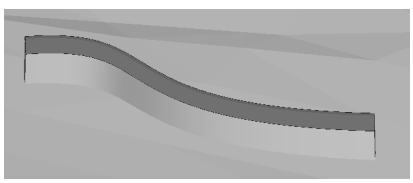
Bézier Landscape Walls

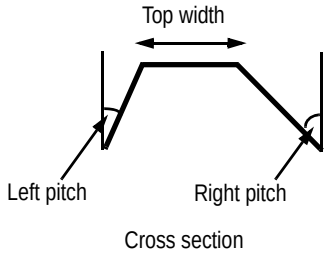
To insert a Bézier landscape wall:

1. Click the **Landscape Wall Bézier** tool from the Site Planning tool set.
2. Click and drag to define the length and angle of the landscape wall. Click again to set the end point. If this is the first time the tool is used during this session, the Object Properties dialog box opens. Accept the default values and click **OK**.
3. Adjust the control points of the Bézier wall by clicking and dragging to obtain the desired curvature.



The Bézier landscape wall parameters can be edited in the Object Info palette.





Click to show/hide the parameters.

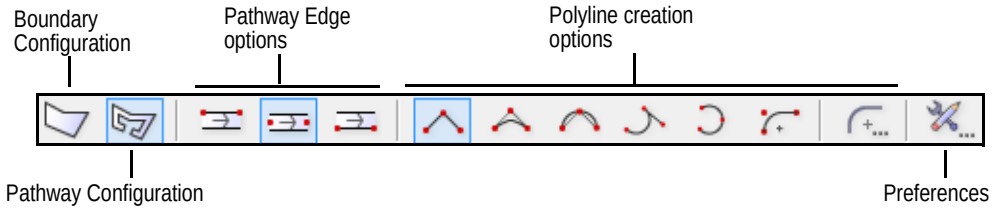
Parameter	Description
Z	Elevation of starting point of wall
Wall Length	Total length of all wall sections
Top Width	Width of wall at top (see diagram)
Wall Height	Wall height above Z value
Resolution	Select a 3D display resolution; low resolution displays fastest, but high resolution provides the best quality
Rise	Difference in height between wall starting and ending points
Left/Right Pitch	Angle of wall on left and right sides (see diagram); for a Bézier wall with site modifiers, the left side is always the side that retains earth
Show Joints	Displays the connecting joints between wall sections
Use Site Modifiers	Adds a pad and grade limits to each wall section and allows the landscape wall to modify the site model
Use Grade Limits	Uses grade limits to define the site modifier modification area
Left/Right Grade Limits Off.	Controls the grade limit offsets on either side of the wall (applies only when site modifiers are used)

Creating Stepped Walls

Creating Hardscape Objects

A hardscape object is comprised of paved areas with joint patterns and optional borders. A boundary hardscape or pathway hardscape can be created. To draw a hardscape object, either use the **Hardscape** tool, or create a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277). The hardscape object can modify the site model.

The following modes are available.



Mode	Description
Boundary Configuration	Creates a hardscape with a boundary configuration, defining an area with an optional border
Pathway Configuration	Creates a hardscape along a path
Pathway Edge options	For the pathway configuration, sets where to create the pathway in relation to the drawn polyline
Polyline creation options	Selects the method for drawing the polyline upon which the hardscape object is based; see “Creating Polylines” on page 298.

Mode	Description
Preferences	Sets the default preferences for the hardscape object

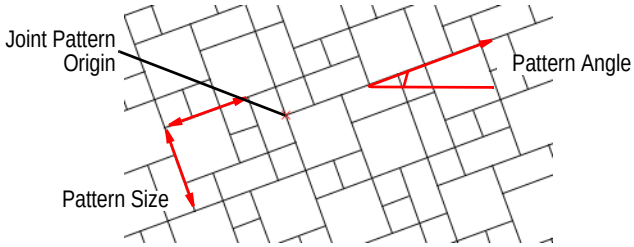
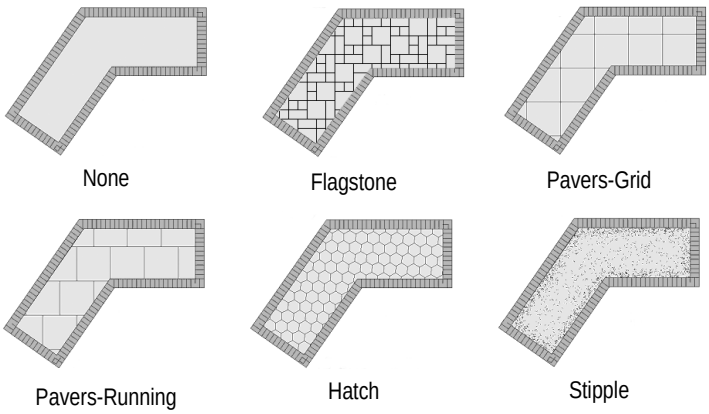
 To create a hardscape object:

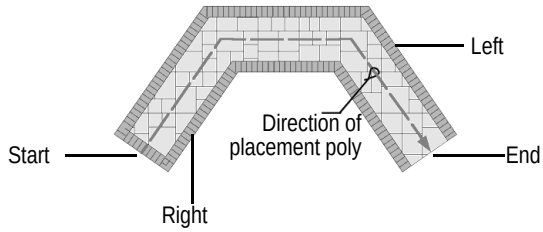
1. Click the **Hardscape** tool from the Site Planning tool set.
2. Click **Preferences** from the Tool bar to specify the default **Hardscape** tool parameters for this session.

   The Hardscape Object Settings dialog box opens. Enter the hardscape parameters.

Click to show/hide the parameters.

Parameter	Description
Definition	
Name	Enter a name for the hardscape object, which displays as the ID in the ID tag
Tag Display	Select whether to display the hardscape tag to the right or left of the leader line, without a leader line (floating), or not at all
Tag Style	Select the ID tag display style. • Select Name Only to display the name of the hardscape object only (as defined in the **Name** field) • Select Name-Area(s) to display the hardscape object name, area and, if applicable, the area of the border • Select Name-Area(s)-Perim to display the hardscape object name, perimeter length and, if applicable, the area of the border.
Tag Class	Select a class for the hardscape tags, to control the tag appearance (line and marker style) and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Hardscape Class> which places the tag in the same class as the hardscape object.
Display Tag Line Marker	Displays a marker at the end of the leader line; specify the marker style by editing the hardscape tag class (see “Setting Class Properties” on page 179)
Snap Tag to Hardscape Edge	Select this option to automatically align the endpoint of the tag with the edge or corner of the hardscape that is nearest to the shoulder point of the tag

Parameter	Description
Joint Pattern	Select the main hardscape joint pattern. The Set Joint Pattern Options dialog box opens when Flagstone, Pavers-Grid, or Pavers-Running is selected. Enter the pattern size; click the **Lock** button to create a symmetrical pattern, or deselect the **Lock** button to create an asymmetrical pattern. Specify the pattern angle.  Alternatively, select a hatch, stipple, or tile resource from either the default content or the current file's content (see "Resource Libraries" on page 219 or "Defining Hatches" on page 1105). Click **OK**. For hatch, stipple, or tile fills, the selected resource's name displays beneath the **Joint Pattern** list.  The joint pattern origin is editable and has a control point marked with a red locus for identification.
Area Class	Select a class for the joint pattern, to control its appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, or select the default class, Site-Hardscape Comp-Main Joint.
Configuration	Sets the boundary and pathway parameters for the hardscape object
Boundary/ Pathway	**Boundary** defines the hardscape object with a user-defined polyline as the outer edge of the hardscape object (for example, to define regular or irregular patio areas), while **Pathway** defines the hardscape object with a user-defined polyline as an alignment for a linear path 

Parameter	Description
Width	Width of the pathway hardscape object
Offset	Sets the offset of the pathway centerline from the placement line
Draw Border	Select to include a border pattern along the edges of the hardscape object
Width	Enter the width of the border
Joint Pattern	Select the border paving pattern. The Enter Value dialog box opens when Spaced Joints is selected; enter the desired joint spacing value. Alternatively, select a hatch, stipple, or tile resource from either the default content or the current file's content. Click **OK**. For hatch, stipple, or tile fills, the selected resource's name displays beneath the **Joint Pattern** list. Similar to the main hardscape joint pattern, the border hardscape joint pattern origin is editable. It has a control point marked with a green locus for identification.
Background Color	Click the color box to select the desired border color
Border Class	Select a class for the border joint pattern, to control its appearance and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, or select the default class, Site-Hardscape Comp-Border Joint.
Pathway Borders	Select the border configuration for pathway hardscape objects with borders 
Draw 3D	Specify 3D hardscape object parameters
3D Type	The hardscape object can act as a 3D slab object or as a pad or texture bed site modifier for a site model. Select the type of 3D modifier to create. Select Slab or Pad Modifier if the paving area is flat. Select Texture Bed Modifier if the paving area covers an area of the site with topographical changes. For more information, see "Site Model Modification Overview" on page 705. Select None to not display the hardscape object in 3D views.
Thickness	For a slab or pad, enter the thickness parameter
Main Texture	Select the texture resource for the hardscape object or assign it by class; the displayed textures are from either the default content or the current file's content. The texture origin and rotation are automatically set to match the 2D fill origin and rotation for the main texture area.
Border Texture	Select the texture resource for the border object or assign it by class; the displayed textures are from either the default content or the current file's content. The texture origin and rotation are automatically set to match the 2D fill origin and rotation for the border.

The **Stipple** joint patterns are a processor-intensive action for large hardscape objects and can significantly increase regeneration time.

When drawing a hardscape object with a curved boundary, speed the regeneration time by setting the **2D conversion resolution** in Vectorworks preferences to low (see "Edit Preferences" on page 49).

3. Click **OK**.

4. Click the appropriate mode in the Tool bar to specify the creation method of the hardscape object. Select whether to create a boundary or pathway hardscape object.

   The hardscape configuration can also be changed after creation from either the Hardscape Object Settings dialog box or the hardscape context menu commands (see “Converting the Hardscape Object to a Boundary or Pathway Configuration” on page 751).

   For information on the **Polyline** tool modes, see “Creating Polylines” on page 298.
5. Click to set the hardscape object’s start point.
6. Click to set the end of the segment and the beginning of the next. Continue drawing segments in this manner until the hardscape object is complete. Return to the start point (for boundary configurations), or simply double-click (for either pathway or boundary configurations) to finish creating the hardscape object.
7. When classing the subcomponents of the hardscape object, such as for the joints or tags, the class visibility of the specified class controls the visibility of the corresponding subcomponent. The class attributes are only applied to the corresponding subcomponent when the **Use at Creation** option is selected for the class. For example, if the Joint Class is set to the Site-Hardscape Comp-Main Joint class, and that class specifies a hatch Fill Style, edit the class and select the **Use at Creation** option to apply the fill attribute.
8. If the hardscape **3D Type** is set to Pad Modifier or Texture Bed Modifier, update the site model to apply the modification. Select the site model and click **Update** from the Object Info palette.

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing the Hardscape Object Settings
Editing Hardscape Object Fills
Applying Colors
Updating a Site Model
Setting Class Attributes

Editing the Hardscape Object Settings

The parameters can be edited for selected hardscape objects through the **Hardscape Settings** button on the Shape tab of the Object Info palette. To modify the default hardscape object settings, click the **Preferences** button on the Tool bar.

Many of the parameters are identical to those used to create the hardscape object (see “Creating Hardscape Objects” on page 745). However, certain parameters are accessible in the Object Info palette only.

Click to show/hide the parameters.

Parameter	Description
Shape Tab	
Hardscape Settings	Click to edit the selected hardscape object. If more than one hardscape object is selected for editing, the **Hardscape Settings** button appears dimmed; select a single hardscape object to access the Hardscape Object Settings dialog box for editing parameters.
Save Hardscape	Click to save the selected hardscape object as a symbol; for more information, see “Saving the Hardscape Object Settings” on page 752. If more than one hardscape object is selected for editing, the **Save Hardscape** button appears dimmed; select a single hardscape object to save it as a symbol.
Definition	
Hardscape Name	Specifies the name assigned to the hardscape object
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Draw Border	Includes a border pattern along the edges of the hardscape object
Border Width	Enter the width of the border
Path Width	Width of the pathway hardscape object
Path Offset	Sets the offset of the pathway centerline from the placement line
Joint Pattern Size X/Y	Specifies the size of the joint pattern units, for flagstone, pavers-grid, and pavers-running joint patterns
Joint Pattern Angle	Specifies the rotation angle of the joint pattern
Border Pattern Size	Sets the width of the spaced joints, when spaced joints are selected for the border
Border Pattern Angle	Enter the angle of the pattern inside the border
Main Unit Price	Sets the price per square unit for the main hardscape material, for use in worksheets
Main Price Code	Specifies the main hardscape price code
Border Unit Price	Sets the price per square unit for the border material, for use in worksheets
Border Price Code	Specifies the hardscape border price code
Tag	
Tag Display	Select whether to display the hardscape tag to the right or left of the leader line, without a leader line (floating), or not at all
Tag Style	Select the ID tag display style. • Select Name Only to display the name of the hardscape object only (as defined in the **Name** field) • Select Name-Area(s) to display the hardscape object name, area and, if applicable, the area of the border • Select Name-Area(s)-Perim to display the hardscape object name, perimeter length and, if applicable, the area of the border.
Tag Class	Select a class for the hardscape tags, to control the tag appearance (line and marker style) and visibility. The classes present in the drawing are listed; alternatively, create a new class by clicking New, or select the class named <Hardscape Class> which places the tag in the same class as the hardscape object.
Tag Approach Angle	Specifies the angle of the leader line, from 0 to 360°
Tag Shoulder Angle	When a tag shoulder line is enabled, sets the angle of the shoulder line, from 0 to 360°
Tag Text Alignment	Select whether the tag text is aligned on top of a continuation of the leader line, or centered vertically at the end of the leader line
Snap Tag to Hardscape Edge	Select this option to automatically align the endpoint of the tag with the edge or corner of the hardscape that is nearest to the shoulder point of the tag
Display Tag Line Marker	Displays a marker at the end of the leader line; specify the marker style by editing the hardscape tag class (see “Setting Class Properties” on page 179)
Draw 3D	

Parameter	Description
3D Type	The hardscape object can act as a 3D slab object or as a pad or texture bed site modifier for a site model. Select the type of 3D modifier to create. Select Slab or Pad Modifier if the paving area is flat. Select Texture Bed Modifier if the paving area covers an area of the site with topographical changes. For more information, see "Site Model Modification Overview" on page 705. Select None to not display the hardscape object in 3D views.
3D Thickness	For a slab or pad, enter the thickness parameter
Information	
Main Area	Displays the size of the main area
Border Area	Displays the size of the border area
Footprint Area	Displays the area of the entire footprint of the hardscape
Main Perimeter	Displays the perimeter measurement of the main area polyline
Border Perimeter	Displays the perimeter of the border only
Footprint Perimeter	Displays the perimeter measurement of the entire footprint polyline
Vertex parameters	Edits the vertices of the path object that the hardscape is based upon; see "Editing Vertex-Based Objects" on page 1002
Render Tab	
Texture	If a **3D Type** was selected in the Hardscape Object Settings dialog box, the Texture drop-down list is enabled for the main area texture (border textures must be set in the Hardscape Object Settings dialog box)

To set individual border segments to invisible for boundary hardscapes or cut-out holes, select the hardscape object, and then click the **Reshape** tool. Select Hide or Show Edges mode. Click the midpoint of the hardscape border segments or the cut-out hole to hide. Repeat this process to set the border segments back to visible, if necessary.

To quickly determine the left and right side of a pathway hardscape object, select the hardscape object and click the middle button next to the **Vertex** field on the Shape tab of the Object Info palette; while the button is clicked, the first vertex of the hardscape object is highlighted with a red box.

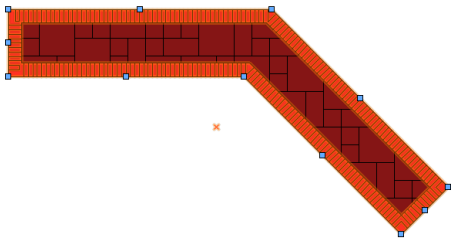
When classing the subcomponents of the hardscape object, such as for the joints or tags, the class visibility of the specified class controls the visibility of the corresponding subcomponent. The class attributes are only applied to the corresponding subcomponent when the Use at Creation option is selected for the class. For example, if the Joint Class is set to the Hardscape - Component - Main Joint class, and that class specifies a hatch Fill Style, edit the class and select the Use at Creation option to apply the fill attribute. See "Setting Class Attributes" on page 181.

Reshaping the Hardscape Object

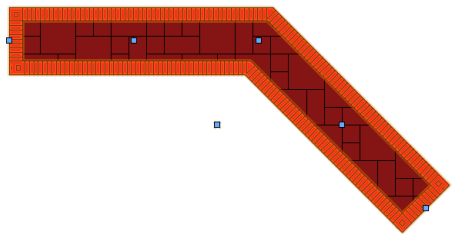
Double-click the hardscape object to activate the **Reshape** tool, or select the **Reshape** tool from the Basic palette. Select the object handles to reshape the hardscape object. For more information, see "Reshaping Objects" on page 1043.

Converting the Hardscape Object to a Boundary or Pathway Configuration

After creation, a boundary hardscape object can be converted to a pathway hardscape object, and similarly, a pathway configuration can be converted to a boundary configuration. The conversion preserves the appearance, but the path of the object changes.



Boundary hardscape object (closed path)



Pathway hardscape object (open path)

To convert a boundary hardscape object to a pathway hardscape configuration:

1. Select the boundary hardscape object.
2. Right-click (Windows) or Ctrl-click (Mac) on the hardscape object and select **Convert to Pathway** from the context menu.

To convert a pathway hardscape object to a boundary hardscape configuration:

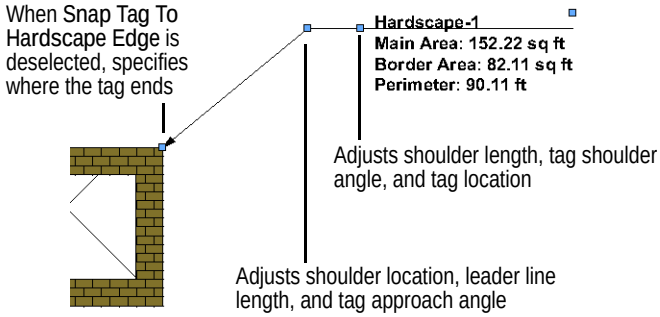
1. Select the pathway hardscape object.
2. Right-click (Windows) or Ctrl-click (Mac) on the hardscape object and select **Convert to Boundary** from the context menu.

   Alternatively, click **Hardscape Settings** from the Object Info palette of a selected hardscape object, and select the configuration.

Hardscape Tag Appearance

Hardscape tags can be adjusted in several ways.

- When hardscape tags are required, their default appearance is normally specified in the hardscape object settings.
- Individual hardscape tags can then be changed for selected hardscape objects by adjusting the hardscape tag parameters in the Object Info palette.
- The hardscape tag class controls the appearance of the leader/shoulder lines, as well as the marker style.
- To align hardscape tags for improved readability, use the **Align/Distribute Leader Lines** command (see “Aligning and Distributing Leader Lines” on page 1035).
- If an individual tag needs to be repositioned, hardscape tags also have two control points for adjusting the shoulder length, tag shoulder angle, and tag location; or the shoulder location, leader line length, and tag approach angle.



Saving the Hardscape Object Settings

Once the hardscape object is set to the desired appearance, the settings can be saved for future use or importing into other files, by saving the hardscape object. When you insert a hardscape from the Resource Browser, all its object settings are preset.

To save the selected hardscape object settings:

1. Select a hardscape object.
2. In the Object Info palette, click **Save Hardscape**.

   The Enter String dialog box opens.
3. Enter a unique name.
4. Click **OK**.

   The hardscape object is saved in the "Hardscapes" symbol folder in the Resource Browser. Boundary and pathway hardscape objects are assigned unique thumbnail view icons for easy identification. For more information on the Resource Browser, see "Using the Resource Browser" on page 221.
5. To use a saved hardscape object, double-click on it in the Resource Browser. Alternatively, drag the saved hardscape onto an existing hardscape to apply the saved settings.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Hardscape Objects
Editing Hardscape Object Fills

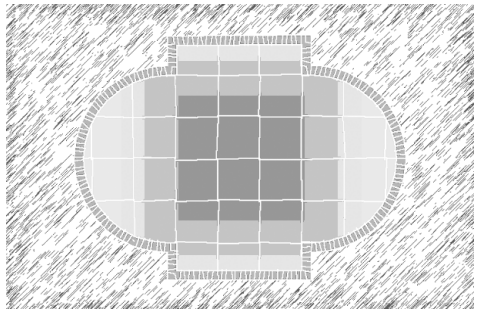
Editing Hardscape Object Fills

Hardscape object fills can be changed through the Attributes palette. If a joint pattern was specified, the joint pattern is drawn over the fill attribute of the hardscape object. The main joint pattern origin is editable and has a control point marked with a red locus for identification; the border joint pattern uses a green locus.

To change the appearance of a hatch fill, edit the hatch definition as described in "Editing Hatch Definitions" on page 1111. Change the size or angle of a hatch fill by temporarily switching to a Pavers-Running pattern, setting the **Pattern Size X, Y** and/or **Pattern Angle** values, and then switching back to the selected hatch.

Hardscape objects can also use images and gradients as fills. Use the **Attribute Mapping** tool to adjust the fill direction and size. For more information on using image or gradients, see "The Attributes Palette" on page 1093; for more information on using the **Attribute Mapping** tool, see "Mapping Fills with the Attribute Mapping Tool" on page 1123.

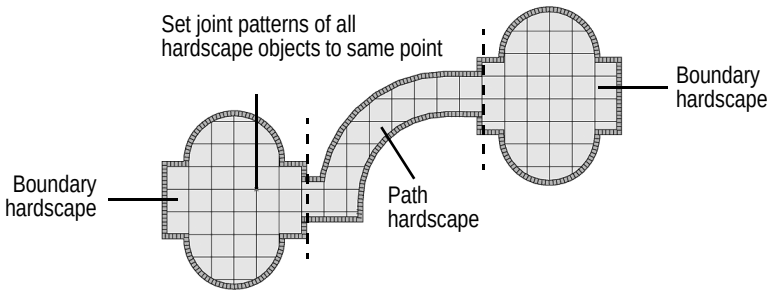
If the subcomponents of the hardscape object are classed, such as for the joints or tags, the class visibility of the specified class controls the visibility of the corresponding subcomponent. The class attributes are only applied to the corresponding subcomponent when the **Use at Creation** option is selected for the class. For example, if the Joint Class is set to the Hardscape - Component - Main Joint class, and that class specifies a hatch Fill Style, edit the class and select the **Use at Creation** option to apply the fill attribute. See "Setting Class Attributes" on page 181.



The angle or origin point of a hardscape fill can be adjusted. Move the control point at the center of the hardscape to control the origin point of the paving pattern (including a hatch) and change the **Joint Pattern Angle** value to adjust the angle.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

When joining bordered hardscape objects with similar borders, use the **Add Surface** command (see "Add Surface" on page 1082).

To coordinate the joint patterns of several hardscape objects, configure the hardscape objects with the same settings and drag their main pattern origin control points (marked with the red locus) to a common location.



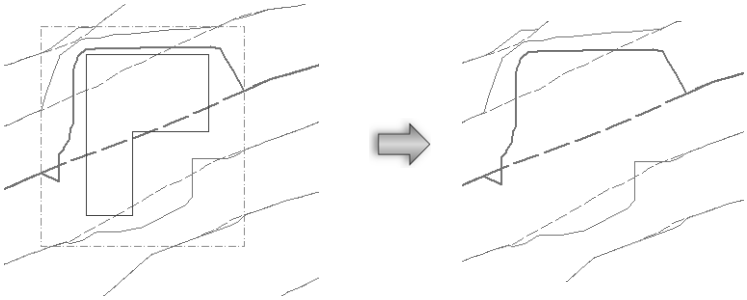
Creating Hardscape Objects

Showing and Hiding Site Modifiers

In the Vectorworks Landmark product, site modifiers such as pads, grade limits, and portions of roads and landscape walls can be made hidden or visible. The modifications to the proposed site model are still visible even when the modifier is hidden.

To show or hide site model modifiers:

1. Select **View > Show > Show or Hide Site Modifiers**.
2. If site modifiers were hidden, they are displayed; if they were visible, they become hidden.



Correcting Site Modifier Errors

If site modifier errors are detected when the site model is updated, an icon displays at the location of each error.

The following modifier problems can generate errors:

- Pad which does not lie inside the grade limits
- Pad intersecting grade limits or the site border
- Two intersecting pads
- Spoil pile intersecting the site border
- Spoil pile intersecting grade limits or the site border
- Two intersecting spoil piles
- Texture bed intersecting the site border
- Two intersecting texture beds
- Locus which does not lie inside the grade limits

If these errors exist, the site model can still be modified; however, results may not be as expected. It may be better to resolve the errors and update the site model again before proceeding.

When the site model object is selected, the Object Info palette's **Error Count** field displays the number of errors, and an icon displays at the location of each error. Hold the cursor over an icon to display a SmartCursor cue that describes the error. Resolve the errors by making adjustments to prevent all intersecting modifiers.

To hide the icons from view (but not resolve the underlying errors), deselect **Show Errors** on the site model's Object Info palette.

Update the site model to ensure that all errors have been corrected.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Validating 3D Source Data
Site Model Properties

A L Drawing Property Lines

Use the **Property Line** tool to draw property lines. Alternatively, use one of the following methods.

- Draw a polyline and then select the **Create Objects from Shapes** command (see "Creating Objects from Shapes" on page 277).
- Draw arcs and lines, select **Modify > Compose** to create a single polyline, and then select the **Create Objects from Shapes** command.

  Click here for a video tip about this topic (Internet access required).

Property Line Tool

The **Property Line** tool interactively creates property boundaries from surveyor's descriptions. The resulting polyline is composed of line and/or arc segments. The closing error can be automatically drawn and measured. Each segment can be individually labeled with distance and bearing; a curve data worksheet displays curve data.

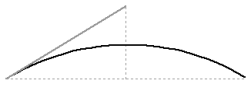
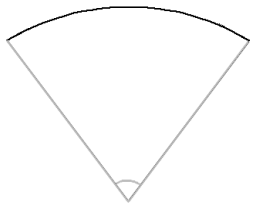
 To create property lines using the **Property Line** tool:

1. Click the **Property Line** tool from the Site Planning tool set.
2. Click to set the starting point of the first property segment.

   A red bull's-eye is placed on the drawing to mark the starting point; the Define Property Line dialog box opens.
3. Specify the segment parameters and click **Add** to update the drawing file; the bull's-eye cursor moves to the end of the segment. Continue to add or remove segments as needed.

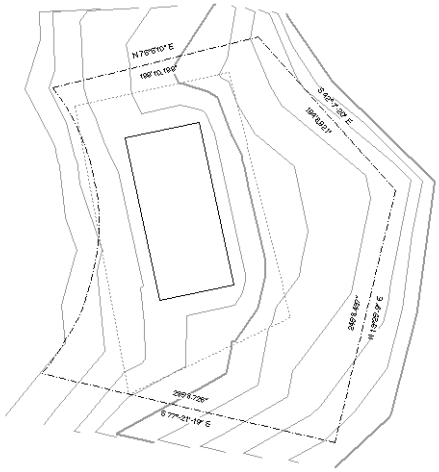
Click to show/hide the parameters.

Parameter	Description
Line	Specify whether the current segment is a line
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Bearing	Enter the azimuth or bearing of the line segment; if entering a bearing, a variety of formats are supported. Use spaces (N 10 30 0 E), degrees—d for degrees, m for minutes, and s for seconds (N10d0m0s E), or decimal degrees (N 10.5d 0m 0s E).
Distance	Enter the distance of the line segment
Curve	Specify whether the current segment is a curve
Start Tangent to Previous	Forces the back tangent to be collinear with the previous segment
Back Tangent	Enter the azimuth of the back tangent of this curve; a variety of formats are supported. Use spaces (10 30 0), degrees—d for degrees, m for minutes, and s for seconds (10d0m0s), or decimal degrees (10.5d 0m 0s).
Radius	Enter the radius of the curve segment
Arc Dist	Select to use the distance along the arc 
Chord Dist	Select to use the distance between the point of tangency and the point of curvature 
Tangent Dist	Select to use the distance from the end point of the last segment to the point of the intersection of the curve 
Delta Angle	Select to use the central angle of the curve as measured from the center of the arc 
Chord Bear.	Select to use the bearing of the arc chord
Forward Tangent	Displays the forward tangent value
Counter-Clockwise	Select to draw the curve **Counter-Clockwise**
Remove	Click to remove the selected segment from the property line
Update	Click to update the parameters for the current segment
Add	Click to add a segment to the property line
< Previous / Next >	Click to move backward and forward through the segments
Automatically create closing segment	Select to automatically create the final segment between the last point and the first

4. Click **OK** to create the property line.

   If this is the first time the **Property Line** tool has been used, the Object Preferences dialog box opens. Accept the defaults and click **OK**.



Editing Property Lines

Editing Property Lines

Edit property lines by using the **Reshape** tool, or by clicking the **Edit with Dialog** button in the Object Info palette. Additional parameters can also be modified in the Object Info palette. See “Reshaping Objects” on page 1043 for information on using the **Reshape** tool for editing.

To edit property lines using the Object Info palette:

1. Select the property line.
2. Modify the parameters in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file’s content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See “Using Text Styles” on page 393 and “Formatting Text” on page 389.
Name	Enter a property line name
Number	Enter a property line number
Area	Displays the area of the property line
Perimeter	Displays the length of the property line perimeter
Show Name	Select to display the property line name
Show Number	Select to display the property line number
Show Area	Select to display the area of the property line, along with a leader line
Area Units	If using Imperial drawing units, select Acres or Square Feet; if using metric drawing units, select Hectares or Square Meters
Decimals	Indicates the number of decimal places to display for the property line area
Text Rotation	Specifies the text rotation degrees

Parameter	Description
Fill Behind Text	Select to display a fill behind text
Annotate Segments	Select to annotate each segment of the property line
Annotation Style	When **Annotate Segments** is selected, indicate the annotation information to display
Angle Format	Select whether the annotations display in Azimuth or Bearing angle format
Show as Texture Bed on Site Model	Select to display the property line as a texture bed on the site model (see "Creating a Texture Bed" on page 715)
Texture Bed Class	If **Show as Texture Bed** was selected, specify the texture bed class
Edit with Dialog	Click to display the Define Property Line dialog box for editing the property line; see "Drawing Property Lines" on page 755 for information on parameters
Vertex parameters	Edits the property line path vertices; see "Editing Vertex-Based Objects" on page 1002

For information on editing object vertices, see "Reshaping Objects" on page 1043.

3. If the property line was designated to be a texture bed on the site model, select **Tools > Organization**. On the Classes tab, select the class designated as the texture bed class, and assign it a distinctive fill color, or, if the Renderworks product is installed, a texture. Select the site model and click **Update** from the Object Info palette; the property line displays as a texture bed on the site model. Switch to a 3D view and render for the full effect.
4. When **Annotate Segments** is selected and curve data exists, a curve data worksheet is automatically created, showing curve data in worksheet form. Select **Window > Worksheets** to display the worksheet, or view it from the Resource Browser.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Property Line Tool

AL Analyzing the Site Model

Reporting Site Model Volume

The individual cut and fill calculations for a site model can be listed in a report for verification.

To list site model volume:

1. Select the site model to analyze. Only one site model can be analyzed at a time.
2. Select the **Create Site Volume List** command from the appropriate menu:
   - Architect workspace: **AEC > Terrain > Create Site Volume List**
   - Landmark workspace: **Landmark > Create Site Volume List**

   The Create Site Model Volume List dialog box opens. On the Columns tab, select the columns of site model data to include in the report.

Click to show/hide the parameters.

Parameter	Description
Available Columns	Lists the columns that can be selected for inclusion in the report
Selected Columns	Lists the columns to be included in the report; to change the list order, use the cursor to drag and drop the items to move in the # column.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Add >	Adds one or more selected item(s) from the available list to the end of the Selected Columns list
Add All >>	Adds all available columns to the end of the Selected Columns list
< Remove	Removes one or more selected item(s) from the Selected Columns list
<< Remove All	Removes all column items from the Selected Columns list
Column Parameters	
Triangle Index	Creates a numeric index for each triangle that makes up the site model Triangulated Irregular Network (TIN)
Existing Surface Area	Lists the surface area of each triangle in the existing site model
Proposed Surface Area	Lists the surface area of each triangle in the proposed site model
Projected Area	Lists the projected area of the common (existing and proposed) site model triangles
Existing Volume	Lists the volume of each triangle (extruded to the existing site model's **Minimum Elevation** plus its **Datum Elevation**)
Proposed Volume	Lists the volume of each triangle (extruded to the proposed site model's **Minimum Elevation** plus its **Datum Elevation**)
Cut Volume	Compares the existing and proposed volume for each triangle in the site model, and displays the extra volume present in the proposed model
Fill Volume	Compares the existing and proposed volume for each triangle in the site model, and displays the volume removed from the proposed model
Net Cut & Fill Volume	Lists the fill volume minus the cut volume for each triangle in the site model
Total Cut & Fill Volume	Lists the fill volume plus the cut volume for each triangle in the site model

3. On the Grade Limits Modifiers tab, select the grade limits site modifiers to include in the report calculations.

Click to show/hide the parameters.

Parameter	Description
Available Grade Limits Modifiers	Lists the grade limit-type site modifiers in the drawing; grade limits modifiers are identified by an assigned number
Selected Grade Limits Modifiers	Lists the site modifiers to be included in the report calculations
Add >	Adds one or more selected item(s) from the available list to the end of the Selected Grade Limits Modifiers list
Add All >>	Adds all available modifiers to the end of the Selected Grade Limits Modifiers list
< Remove	Removes one or more selected item(s) from the Selected Grade Limits Modifiers list
<< Remove All	Removes all items from the Selected Grade Limits Modifiers list
Show Modifier	When a numbered grade limits modifier is highlighted, selects the associated site modifier in the drawing, and zooms in so the site modifier is easily identifiable

Parameter	Description
Modifier Parameters	
Outside Grade Limits	Includes all areas of the site model surface without any modifiers
Grade Limits Modifier __	Each grade limits site modifier in the drawing is identified with a number. If you have provided a name for the site modifier on the Data tab of the Object Info palette, the name displays in parentheses

4. On the Options tab, choose report output parameters.

Click to show/hide the parameters.

Parameter	Description
Show Data for Unmodified Site Model Surface	When selected, the report includes the entire surface of the site model. Deselect this option to exclude the unchanged surface between the existing and proposed site models from the report.
Create Text File	Creates a tab-delimited text file of the report, which can easily be imported into a spreadsheet program for further manipulation. Specify the name of the text file and the location to save it in.
Create Triangle Group	Creates a group containing subgroups of polygons and volumetric data text for each triangle that makes up the site model triangulated network. For complex site models, a significant amount of information is displayed; adjust text size of the entire selected group with **Text > Size**.
Create Worksheet	Adds a Volume List worksheet containing the volume data to the current file

5. Click **OK** to create the site model volume report.

Zone of Visual Influence Analysis

This type of analysis creates a representation of shadowed and lighted areas as seen from a specified point of view.

To perform a zone of visual influence analysis:

1. Select the site model to analyze.
2. Select the **Zone of Visual Influence** command from the appropriate menu:
   - Designer workspace: **AEC > Terrain > Zone of Visual Influence**
   - Landmark workspace: **Landmark > Zone of Visual Influence**.

   The Zone of Visual Influence dialog box opens. Specify the analysis parameters.

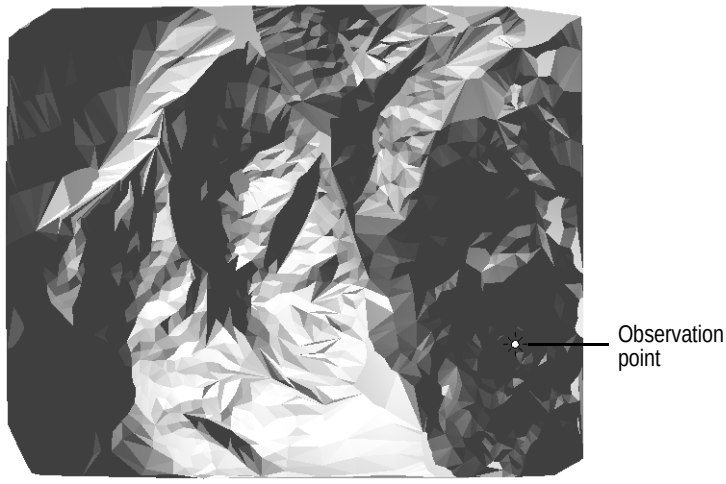
Click to show/hide the parameters.

Parameter	Description
Select Site Model	Select the site model to analyze
View height above Site Model	Represents the view point elevation for the study. For example, 60 inches (152.4 cm) is often used as a standard eye level when standing.

3. Click **OK**.
4. Click on the site model to indicate the observation point.

5. Click again to perform the analysis.

   A light source is automatically inserted, and the layer is rendered to complete the analysis. Dark regions indicate areas that cannot be seen from the specified observation point at that viewing level.



Zone of Visual Influence analysis

Reporting Site Model Volume

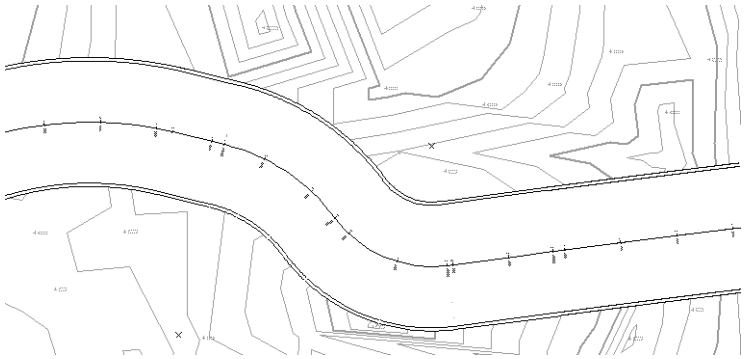
AL Obtaining Site Model Data

Site Model Section

Use the **Site Model Section** command to create a profile or sectional views of the site.

To create a site model section:

1. Draw or select a 2D polygon or polyline to define the section alignment. In the Vectorworks Landmark product, if you are selecting a previously drawn polyline with station points (such as a road or Roadway (NURBS) object), a station point profile can be created.



   The polygon or polyline vertices must be contained within the limits of the site model.

2. With the polygon or polyline selected, select the **Site Model Section** command from the appropriate menu:
   - Architect workspace: **AEC > Terrain > Site Model Section**
   - Landmark workspace: **Landmark > Site Model Section**

   The Create Site Model Section dialog box opens. Specify the site model section parameters.

Click to show/hide the parameters.

Parameter	Description
Create section on layer	Specify the layer where the section will be placed
Draw Background Grid	Select to draw a background grid, and click the color box to specify the grid color
Draw Existing Site Profile	Select to draw a profile for the existing site model, and click the color box to specify the profile color
Draw Proposed Site Profile	Select to draw a profile for the proposed site model, and click the color box to specify the profile color
Draw Selected Station Profile (Vectorworks Landmark required)	Select to draw a profile of the station points, and click the color box to specify the profile color; this option is available only if a polyline associated with station points (such as a road) is selected
Station Points	
Draw points at polygon vertices	If the selected polygon does not contain stake objects, select to create station points at the polygon vertices; this option is available only if a station polyline is selected
Draw points at interval	If the selected polygon does not contain stake objects, select to create station points at the specified interval
Draw points at existing stakes (Vectorworks Landmark required)	When stakes exist along the profile polygon, station points are automatically created for the stakes; this option is available only if a polyline with station points (such as a road) is selected
Formatting	Click to specify the section graphic formatting

3. Click **Formatting**.

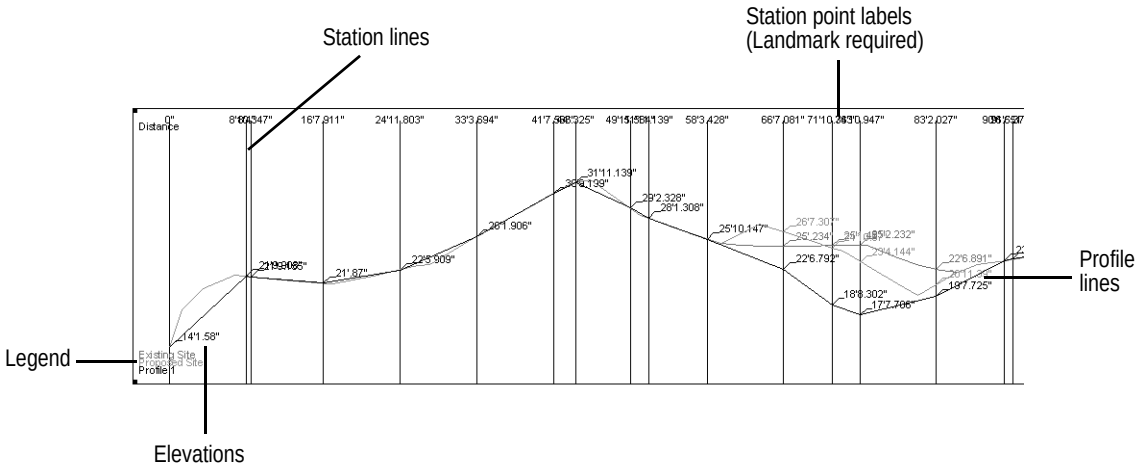
   The Site Model Section Formatting dialog box opens. Specify the formatting for the site model section graphic.

Click to show/hide the parameters.

Parameter	Description
Display Scale Factor	Specifies the site model scale factor; the layer scale factor is entered by default
Vertical Magnification Factor	Sets an independent scale factor for the Y axis of the graphic
Vertical Margin	Specifies the distance, in page units, between the highest and lowest points in the graph and the graph margins
Horizontal Margin	Specifies the distance, in page units, between the graph start and end and the graph margins
Draw Legend Text	Indicates whether to create a legend for the graph; legend text is color coded to match the section profiles
Legend point size	Specifies the size, in points, of the legend text
Draw Elevations	Indicates whether to display the elevation of the profiles along the station points
Elevation point size	Specifies the size, in points, of the elevation text
Station Point Labels (Vectorworks Landmark required)	Sets the type of label, if any, to apply to the station points on the background grid

Parameter	Description
Profile Line Weight	Specifies the thickness of the profile lines
Station Line Weight	Specifies the thickness of the station lines

4. Click **OK** to return to the Create Site Model Section dialog box.
5. Click **OK** to create the sectional view of the site. If the site model changes, the site model section needs to be recreated.



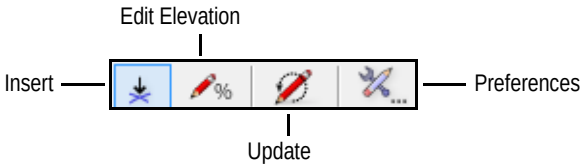
Calculating Grade
Inserting Stake Objects

Calculating Grade

The **Grade** tool annotates terrain slopes on site plan documents.

Networks of grade objects can be created to show slope characteristics across a surface. Grade objects can be used in conjunction with a site model, and can modify the site model. In addition, 3D loci can be generated from grade object elevations, and used as the basis of creating a site model.

The **Grade** tool has three modes:



Mode	Description
Insert	Draws a grade object based on set elevation parameters or a site model
Edit Elevation	Changes the elevation value of one or more existing grade objects
Update	Updates all grade objects in the drawing, applying current preference settings and updating any overlapped elevation points
Preferences	Sets the default global preferences for the grade object

Specifying Global Grade Object Preferences

The grade indicator appearance for all grade objects in the file can be specified globally.

 To set global grade object appearance:

1. Click the **Grade** tool from the Site Planning tool set.
2. Click **Preferences** from the Tool bar. Alternatively, select an existing grade object; click **Settings** from the Object Info palette, double-click the grade object, or right-click (Windows) or Ctrl-click (Mac) on a grade object and select **Edit** from the context menu.

   The Grade Settings dialog box opens. Click the Global Settings tab and enter the desired settings.

Click to show/hide the parameters.

Parameter	Description
Display Settings (global)	
Arrow Line Length	Specifies the length of the grade directional arrow, in page units
Arrow Offset	Sets the distance of the grade directional arrow from the grade line, in page units
Grade Def.	Select the type of display for the slope (percent, permille, run/rise ratio, rise/run ratio, or angle)
Precision	Sets the displayed precision for the slope value
Directional Indicator Attributes (global)	
Pen/Style	Select the pen style and line thickness for the grade directional arrow. Alternatively, select Class style in the **Pen** list. The pen, color, style, and marker are then determined by the class attributes of the grade object.
Color	Click the color box to select the grade directional arrow color
End Marker	Specifies the marker type for the end of the grade directional arrow
Opacity	Sets the opacity of the grade directional arrow; drag the slider to the left to decrease opacity, or enter an opacity percentage
Use Class Opacity	The opacity of the arrow can be set by the class of the grade object
Reshape Setting (global)	
Preserve Grade if one point is not connected	Preserves the grade of a reshaped grade object, and recalculates the elevation. This preference works only if at least one end point of the reshaped grade object is not connected to another grade object; the elevation of the unconnected end point is changed. If both end points of the reshaped grade object are connected to other grade objects, the grade is changed and the elevations are preserved to maintain continuity with the connected grade objects.

Parameter	Description
Analyze Grade Objects	Uses grade objects to analyze the slope or elevation

3. Click **OK** to set the global grade indicator attributes.
4. Click **Update** from the Tool bar to update all grade objects in the drawing with the global settings.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Calculating Grade
Applying Object Attributes
Inserting Grade Objects
Analyzing Grade

Inserting Grade Objects

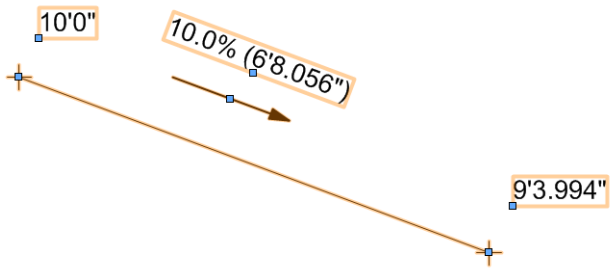
To insert a grade object:

1. Click the **Grade** tool from the Site Planning tool set, and select **Insert** mode from the Tool bar.
2. Click to set the start of the grade. Move the mouse and click again to set the end point.

   The Grade Settings dialog box opens. Click the Object Properties tab and enter the desired settings.

Click to show/hide the parameters.

Parameter	Description
General	Given two parameters, the grade parameters are calculated for the remaining parameters. Enter the two known parameters to calculate and display the other grade parameters. The initial elevation of the first and/or second points of the grade object is determined by these methods: • If an existing grade object is under one of the points, the elevation of the existing grade object is used • If a stake object is under one of the points, its elevation is used • If a site model is under one of the points, the calculated elevation of the point is used • If none of the above occurs, the elevation of the last created grade object is used
Parameter 1	Select the first known parameter (Elevation 1, Elevation 2, Downward Grade in %, Upward Grade in %, Downward Ratio (rise/run), Upward Ratio (rise/run), or elevation change) and enter the value(s) associated with the parameter
Parameter 2	Select the second known parameter (available choices depend on the first parameter selected), and enter the value associated with the parameter
Parameter Display	The remaining parameters are calculated and displayed, including the grade length
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

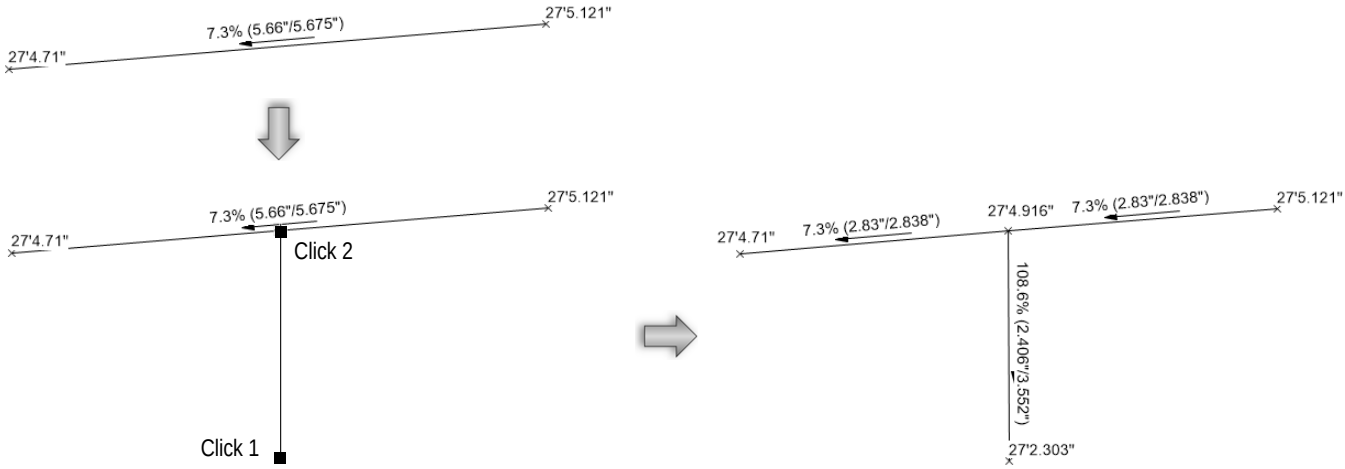
Parameter	Description
Graphics	Sets the graphic appearance of the grade object. The appearance of the indicator, grade definition, and precision is set in the global settings (see “Specifying Global Grade Object Preferences” on page 764). Once created, each graphic element of the grade object has its own control point. Move a control point to adjust the position of the text and indicator. 
Elevation marker symbol	Select the symbol to use to indicate the elevation marker at each end of the grade object; grade symbols are also provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219)
Scale	Scales the elevation marker with respect to the layer scale by the specified scale factor; a **Factor** of less than one decreases the size of the marker while a **Factor** of more than one increases its size
Show Slope Value	Displays the grade value If the value does not display, increase the text size by selecting **Text > Size**.
Show Projected Length/Surface Length Value	Displays the projected grade length/surface length and the slope arrow; the slope arrow always indicates the downward slope direction. Enter a prefix for the value, if desired. The projected length is the distance between the points as measured along a horizontal projection; the surface length is the distance between the points as measured along the surface, which facilitates a more accurate calculation for purchasing materials. If both **Show Projected Length Value** and **Show Surface Length Value** are selected, both values are displayed along the grade object, with a slash (/) in between them (prefix + projected length/prefix + surface length)
Show Elevation Value	Displays the elevation value at each end of the grade object
Draw Line	Draws a slope line to represent the distance between the two elevation points
Site Model	
Grade Object Mode	
Use Grade Object Heights	Uses the elevation values specified in the General parameters
Use Heights from Site Model	When the grade is drawn over a site model, the elevation of the first and second points is derived from either the existing or proposed site model. The values in the General parameters do not apply. The grade object updates with the site model.

Parameter	Description
Change Site Model (Use Grade as Pad Modifier)	Uses the grade object as a site modifier object; the elevation values specified in the General parameters are applied to either the existing or proposed site model. Update the site model to apply the changes (select the site model and click **Update** from the Object Info palette).
Apply To	
Existing/Proposed Site Model	Selects the site model to provide elevation values when **Use Heights from Site Model** is selected, or the site model to be modified when **Change Site Model** is selected

3. Click **OK**.

Grade objects can be edited from the Object Info palette. Commonly-required parameters can be accessed directly from the Object Info palette, or click **Settings** to change any of the parameters of selected grade objects. Alternatively, edit the settings of a grade object by double-clicking on it, or right-click (Windows) or Ctrl-click (Mac) on a grade object and select **Edit** from the context menu.

Grade objects can be used together, using an existing grade object as a reference for another grade object. Either move the endpoint of one grade object onto the line of another grade object, or create a new grade object with an end point on an existing grade object. The existing grade object is split at the new endpoint, creating a network of overlapping grade objects. The elevation of the shared point is interpolated from the slope of the existing object; if an elevation point is updated, all overlapping grade objects automatically update. Click **Update** mode from the Tool bar of the **Grade** tool to force an update of all overlapping grade objects.



Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Calculating Grade
Specifying Global Grade Object Preferences
Editing Elevation
Analyzing Grade
Creating Loci from a Grade Object
Site Model Section
Inserting Stake Objects
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Elevation

If the elevation changes after one or more grade objects have been placed, there is no need to replace the existing grade object(s). Change the elevation of one end of a grade object, or offset the elevation of selected (or all) grade objects in the drawing.

Elevation marker symbols must be at a valid scale to edit elevations. If the Edit Elevation dialog box does not open, increase the marker scale factor for the grade object.

To edit grade object elevation:

1. Click the **Grade** tool from the Site Planning tool set, and select **Edit Elevation** mode from the Tool bar.
2. Click at the start or end of a grade object.

   On the Edit Elevation dialog box, edit the grade as needed.

Click to show/hide the parameters.

Parameter	Description
Elevation at Selected End Marker	Changes the elevation for the selected end of the grade object
Offset Selected Grade Objects	Adjusts the elevation of all selected grade objects by the specified amount
Offset All Grade Objects	Adjusts the elevation of all grade objects in the file by the specified amount

3. Click **OK** to edit the elevation of the grade object(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Calculating Grade
Specifying Global Grade Object Preferences
Inserting Grade Objects
Analyzing Grade
Creating Loci from a Grade Object

Analyzing Grade

Grade objects can be used to analyze the slope objects in a drawing and identify critical slopes. Grade objects that are outside specified slope or elevation ranges are highlighted or selected.

To perform a grade analysis:

1. Click the **Grade** tool from the Site Planning tool set.
2. Click **Preferences** from the Tool bar. Alternatively, click **Settings** from the Object Info palette of a selected grade object.

   The Grade Settings dialog box opens. Click the Global Settings tab and specify the grade analysis parameters.

   Global attribute display of the slope indicator is described in “Specifying Global Grade Object Preferences” on page 764

Click to show/hide the parameters.

Parameter	Description
Analyze Grade Objects	
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Mark Objects	Select the action to take when critical grade objects are identified
Highlight Critical Objects	Click the color box to select the highlight color for grade objects that meet the criteria
Select Critical Objects	Selects all grade objects that meet the criteria
Criteria	Specifies the criteria for critical grade objects
Grade out of range (%)	Sets a range of acceptable slope values; grades outside of the range are considered to be critical
Elevations out of range	Sets a range of acceptable elevation values; elevations outside of the range are considered to be critical
Show intersections	Specifies that any intersecting grade objects should be considered critical

3. Click **OK**. Grade objects that do not meet the criteria are considered critical, and are highlighted and/or selected, as specified.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Calculating Grade
Specifying Global Grade Object Preferences
Inserting Grade Objects
Editing Elevation
Creating Loci from a Grade Object
Setting Site Model Graphic Properties

L Creating Loci from a Grade Object

One or more grade object(s) can be used to place 3D loci on the current layer. The 3D loci can then be used to create stake objects or a site model.

To create 3D loci from stake object elevations:

1. Select one or more grade objects.
2. Select the **Create 3D Loci at Grade Points** command from the appropriate menu:
   - Designer workspace: **AEC > Terrain > Create 3D Loci at Grade Points**
   - Landmark workspace: **Landmark > Create 3D Loci at Grade Points**

   A 3D locus is placed at the ends of each selected grade object. The Z-value of the loci match the associated elevation of the grade start or end point.

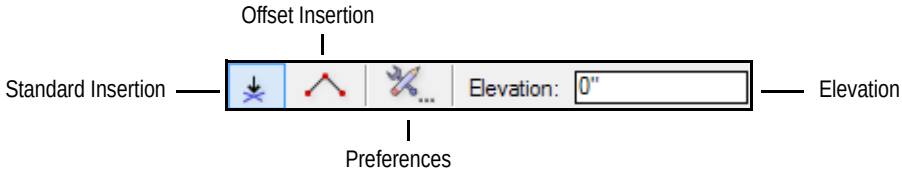
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Calculating Grade
Inserting Grade Objects
Creating the Site Model
Converting 3D Loci to Stake Objects

A L Inserting Stake Objects

Stake objects represent a 3D point in space, with text to display the elevation, coordinate points, or other information. Using a stake object, the elevation information for a point on the site model can be determined and labeled. A stake object can be used as a site modifier. If the design layer is georeferenced, a stake object can be used as a geographic marker.

The following modes are available.



Mode	Description
Standard Insertion	Inserts a single stake object
Offset Insertion	Inserts a stake at each vertex of the drawn polygon shape
Preferences	Sets the default preferences for the stake object
Elevation	Sets the default elevation value for stake objects

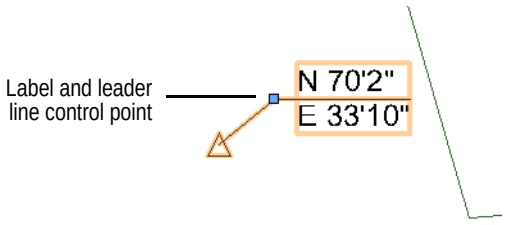
To insert a stake object:

1. Click the **Stake** tool from the Site Planning tool set, and select the mode from the Tool bar.
2. Click on the site model to place the stake object, or draw a temporary polygon to place a series of stake objects at each click point and double-click to end the series.

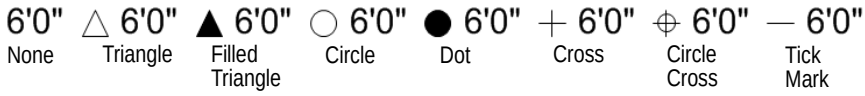
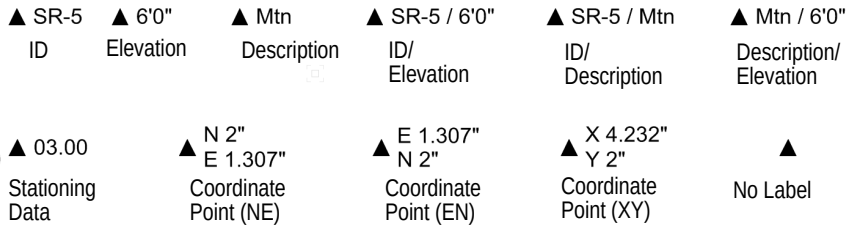
   If this is the first time the **Stake** tool has been used, the Object Properties dialog box opens. Accept the defaults and click **OK**.

   To automatically increment the ID parameter when placing new stake objects, select **Auto ID Numbering** and specify an **Initial ID Number**. Otherwise, leave **Auto ID Numbering** unselected and specify the default **ID Number** for stake objects.
3. The stake objects are added to the drawing at the elevation specified in the Tool bar.

   Select the **Send to Surface** command to set the stake elevation to that of the site model surface (see “Sending Objects to the Site Model Surface” on page 705).
4. If the stake objects are site modifiers, update the site model to apply the changes (select the site model and click **Update** from the Object Info palette.).
5. Move the control point to adjust the label and leader line position. If there are several stake objects, use the **Align/Distribute Leader Lines** command to improve readability (see “Aligning and Distributing Leader Lines” on page 1035).
6. The stake properties can be edited in the Object Info palette.



Click to show/hide the parameters.

Parameter	Description
Z-value	Specifies the stake elevation. To create road stakes with the Vectorworks Landmark product, set the elevation of the first stake along the polyline; the elevation of the remaining stakes is automatically calculated (as described in “Setting Stake Elevations” on page 730).
Rotation	Rotates the stake object indicator and the label (if applicable) around the stake object indicator’s center
Mode	
Include as site model data	Uses the stake as site model data if it is located in the source data layer (select when using stakes to create a site model)
Set elev to site model	Sets the stake elevation to the existing site model elevation
Use as 2D graphic only	Treats the stake like a text box and does not include 3D information such as elevation
Site modifier object	Treats the stake object as a site modifier, which has an effect on the site model
Site Model	When the stake object is used as a site modifier, specifies whether it affects the existing or proposed site model
Style	Selects the graphic style of the stake object indicator 
Mark Offset X	Offsets the stake object indicator from the insertion point along the X axis
Mark Offset Y	Offsets the stake object indicator from the insertion point along the Y axis
Mark Rotation	Rotates the stake object indicator around its center
Label Reference	Selects the type of information to display in the label; select **No Label** to display no text. The available label fields depend on the label reference selected. 
Label Position	Specifies the alignment of the label, and which side to place the leader line (when displayed): • Auto: Automatically determines whether to place the text on the left or right, depending on the position of the label related to the stake’s location • Right: Places the text to the right of the label control point and left-justifies the text • Left: Places the text to the left of the label control point and right-justifies the text • Center: Centers the text on the label control point
Description	For description type labels, enter description text
ID Prefix	For ID type labels, enter a prefix to display before the ID number

Parameter	Description
ID Suffix	For ID type labels, enter a suffix to display after the ID number
Auto ID Numbering	Select to automatically increment the ID number parameter when inserting new stake objects
(Initial) ID Number	For ID type labels, specify the ID number. When **Auto ID Numbering** is selected, the **ID Number** field changes to **Initial ID Number,** and ID numbers will be automatically incremented beginning with the initial number specified. When **Auto ID Numbering** is not selected, specify the default **ID Number** for stake objects.
Station Point Data	For Stationing Data labels, indicates the station point data for the selected stake object
Coordinate Point	Coordinate point type labels can show the stake's distance from a point of origin (user origin), or they can show the stake's geographic position (latitude and longitude) if the design layer is georeferenced (see "GIS and Georeferencing" on page 775). • To display distance, specify Document Units, Feet, or Meters as the **Coordinate Units**. The Coordinate Point (NE - Northing-Easting) and Coordinate Point (EN - Easting-Northing) options display the distance prefixed with the direction (N, W, S, E). The Coordinate Point (XY) option displays the distance prefixed with the file coordinates (X, Y). • To display longitude and latitude, specify Decimal Degrees or Degrees/Minutes/Seconds as the **Coordinate Units**. If the stake object or user origin moves, the coordinates automatically update to reflect the change.
Coordinate Units	For coordinate point type labels, specifies the units to use for the coordinates. • Select Document Units, Feet, or Meters to show the stake's position relative to the user origin of this drawing. • If the design layer is georeferenced, select Decimal Degrees or Degrees/Minutes/Seconds to show the stake's geographic position.
Coordinate Prec	When coordinate point type labels are set for units of Feet, Meters, Decimal Degrees or Degrees/Minutes/Seconds, specifies the precision with which the measurements will be labeled
Show Elevation	For coordinate point type labels, select to display the elevation below the other coordinates
Elevation Units	Specifies the units to use for the elevation. • Select Document Units, Feet, or Meters to show the stake's position relative to the user origin of this drawing. • If the design layer is georeferenced, select Decimal Degrees or Degrees/Minutes/Seconds to show the stake's geographic position.
Elevation Prec	For coordinate point type labels where the elevation value was selected to display, and the specified elevation units are Feet, Meters, Decimal Degrees or Degrees/Minutes/Seconds, specifies the precision with which the measurements will be labeled

Parameter	Description
Show Unit Mark	Select to display the unit mark along with the unit value If Document Units are set for **Coordinate Units**, the unit mark visibility is determined by the document setting.
Display Leader Line	Adds a leader line from the label to the stake object (adjust the leader line position by moving the leader line control point)
Scale Factor	Determines the size of the stake object indicator

Converting 3D Loci to Stake Objects
Site Model Section
Calculating Grade
Creating the Site Model
Updating a Site Model
Aligning and Distributing Leader Lines

Converting 3D Loci to Stake Objects

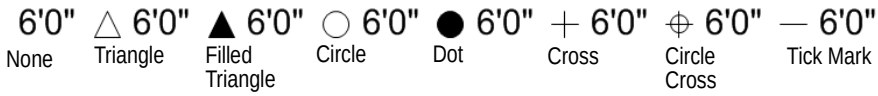
3D loci can be converted to stake objects, which can then be used to modify a site model. This is useful when survey data, in the form of 3D loci, need to modify an existing or proposed site model.

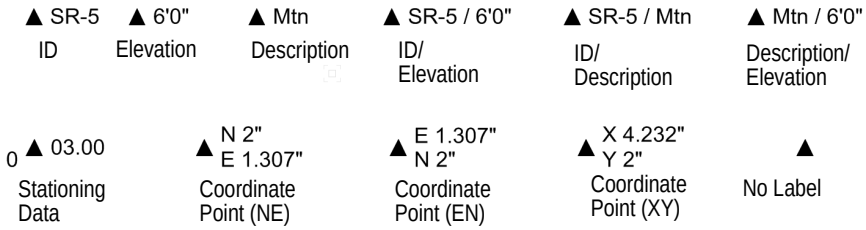
To create stake objects from 3D loci:

1. Select the 3D loci.
2. Select **Modify > Convert > Stake Object from 3D Locus**.

The Convert to Stake Object dialog box opens. Set the properties for new stakes.

Click to show/hide the parameters.

Parameter	Description
Mode	Select the type of stake object(s) to create: • **Include as site model data:** Uses the stake as site model data if it is located in the source data layer (select when using stakes to create a site model) • **Set elev to site model:** Sets the stake elevation to the existing site model elevation • **Use as 2D graphics only:** Treats the stake like a text box and does not include 3D information such as elevation • **Site modifier object:** Treats the stake object as a site modifier, which has an effect on the selected site model
Site Model	Select the site model which the stakes affect
Style	Selects the graphic style of the stake object 

Parameter	Description
Label Reference	Selects the type of information to display in the label; select **No Label** to display no text. The available label fields depend on the label reference selected. 
ID Prefix	For ID type labels, enter a prefix to display with the ID
Auto ID Numbering	Select to automatically increment the ID number parameter when inserting new stake objects
(Initial) ID Number	For ID type labels, specify the ID number. When **Auto ID Numbering** is selected, the **ID Number** field changes to **Initial ID Number,** and ID numbers will be automatically incremented beginning with the initial number specified. When **Auto ID Numbering** is not selected, specify the default **ID Number** for stake objects
Description	For description type labels, enter description text
Station Point Data	For Stationing Data labels, indicates the station point data for the selected stake object
Coordinate Units	Coordinate point type labels show the stake's distance from a point of origin (user origin). This field specifies the units (document units, feet, or meters) to use for the coordinates.
Display Leader Line	Adds a leader line from the label to the stake object (adjust the leader line position by moving the leader line control point)
Scale Factor	Determines the size of the label display
Delete Source Loci	Deletes the original loci that were converted

3. Click **OK** to create the stake objects.

   If the stake objects are site modifiers, update the site model to apply the changes (select the site model and click **Update** from the Object Info palette).

   See "Inserting Stake Objects" on page 769 for additional stake object parameters that can be edited from the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Stake Objects
Updating a Site Model
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

GIS and Georeferencing

In the Vectorworks Architect and Landmark products, there are several features for importing and exporting GIS (geographic information system) data, and for georeferencing Vectorworks drawings.

For a small-scale architectural or landscape project, you might want to accurately import information related to the project site, such as property boundaries or municipal utility resources.

For a larger scale map project, you might import multiple shapefiles and image files and arrange them into maps, adding colors, symbols, and notes based on database information. You might then export files for use in other GIS software.

Both the Vectorworks Architect and Landmark products can store georeferencing information for design layers, as well as import and export georeferencing information related to shapefiles and image files. The Vectorworks Landmark product has additional features to make common GIS operations easier, including tools for adding graticule and great circle objects to a drawing.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~

A L Entering Georeferencing Information for the Document

Georeferencing options are complex; what works for architectural workflows may not be suitable for GIS workflows. Different regions of the Earth have different geographical requirements.

Since the earth is not a perfect sphere, it is not a straightforward process to project a map onto the earth's surface, or transfer an object onto the surface. Projections are mathematical formulas that translate the shape of the earth onto a 2D page. Because the translation loses data, different projections represent the longitudinal and latitudinal coordinates of the non-uniform sphere onto the page.

When georeferencing is enabled, each design layer stores information that describes the mapping of that layer's Cartesian coordinate system onto a geographic coordinate system of latitude and longitude. The origin offset determines how the projection relates to the internal origin (the center of the drawing in the Vectorworks file); this offset can be turned off at the document level or at the layer level for GIS workflows.

If all or most design layers will have the same georeferencing information, use the **Georeferencing** command to set the parameters for the document. After the default document georeferencing is set, edit each design layer to enable georeferencing, and to customize the settings for individual layers, if needed.

Layers without georeferencing will be assumed to be in the Equirectangular projection, which maps latitude and longitude directly onto the X-Y axis. The origin will represent 0°, 0°, and the scale will be equal to that of the Earth's scale of latitude and longitude at the Equator. Some georeferencing tools do not work properly on design layers that are not georeferenced.

To set georeferencing for a document:

1. Select **File > Document Settings > Georeferencing**. The Document Georeferencing dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Adjust origin and orientation of georeferenced data to match document coordinates	Offsets the location of the georeferenced data so that it matches the way that the projection relates to the drawing's internal origin. Normally, enable the option for architectural documents which center and orient around geographical latitude/longitude coordinates; deselect the option when working with raw GIS data (shape files), when no offset is needed, and the center of the data set is also considered to be the center of the projection.
Latitude/Longitude	Specifies the latitude and longitude at the document origin (0,0); enter decimal degrees, or degrees/minutes/seconds (for example, 39.18, 39° 10' 32", or 39d 10m 32s) preceded by a minus (-) sign when appropriate
Angle to True North	Specifies which direction is north from the document origin; use an azimuth degree measurement (0° is along the Y axis, 90° is along the positive X axis)
Projection	Select the appropriate projection type, based on the drawing's scale and purpose; additional parameters that are required to map to the selected projection display in the area below this field. See "Projection Options" on page 783 for details about projections. Alternatively, select the WKT (Well Known Text) option and specify any other projection or coordinate system that is supported by GDAL (Geospatial Data Abstraction Library) in the WKT markup language. For the Vectorworks Architect product, only the Lat/Lon Document Units, Universal Transverse Mercator, and State Plane Coordinate System (NAD83) options are available.
Ellipsoid	Select the appropriate reference ellipsoid used to create the projection (applies to all projections except State Plane Coordinate System NAD83 and NAD27)
UTM Grid Zone/ Hemisphere	For the Universal Transverse Mercator (UTM) projection, select the UTM grid zone and UTM hemisphere that apply to this drawing
State/Zone	For a State Plane Coordinate System (NAD83 or NAD27) projection, select the state and zone that apply to this drawing
Center Latitude/Longitude	Specify the latitude and longitude of the center of the selected projection (applies to Equirectangular, Azimuthal Equidistant, Transverse Mercator, Cassini-Soldner, Gnomonic, or Stereographic projections)
Scale	For the Transverse Mercator projection, specify the scale factor along the central longitude; for the Stereographic projection, specify the scale factor at the central point
Standard Parallel 1/2	For the Lambert Conformal Conic projection, enter the two standard parallels to use for the projection

Parameter	Description
Transformation mode	Select the appropriate option for transforming the layer objects when the layer's projection type is changed: • **Do not transform this layer's geometry:** Layer objects will not be transformed when the layer's projection is changed • **Transform all geometry individually:** Best for projecting geographic data, such as municipal boundaries, roads, rivers, and locations on a map • **Move and rotate all geometry as a group:** Best for projecting non-geographic drawing objects, such as architectural plans
Reset	Discards any edits made and returns all fields to their former values

2. Click **OK** to save the georeferencing settings.
3. Enable georeferencing for each design layer that requires it, as described in "Entering Georeferencing Information for a Design Layer" on page 777.

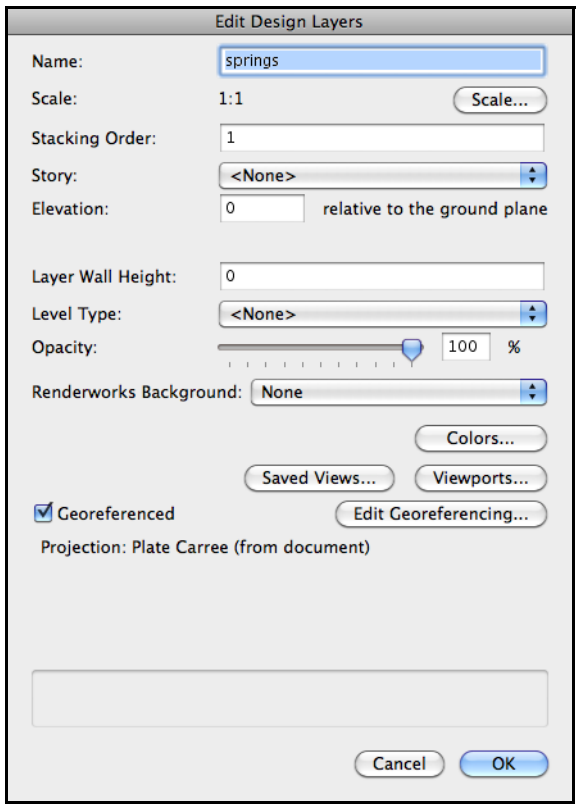
AL Entering Georeferencing Information for a Design Layer

Use the Edit Design Layers dialog box to enable georeferencing for the layers that will use it. Options set here can override the georeferencing options set for the document. Once you have a georeferenced design layer, you can import georeferenced files into it, as well as export images or shapefiles that include georeferencing information.

To accurately measure items on a georeferenced design layer, use the **Great Circle** tool (from the Site Planning tool set), instead of the dimension tools from the Dims/Notes tool set. See "Creating a Great Circle" on page 781.

To enable georeferencing for a design layer:

1. Select **Tools > Organization** to open the Organization dialog box. From the Design Layers tab, select the layer(s) to be changed, and click **Edit**.
2. From the Edit Design Layers dialog box, select **Georeferenced**. The current projection for the layer displays below that field; the default is the current projection setting for the document.



3. To edit the georeferencing information, click **Edit Georeferencing**.

   The Georeferencing dialog box opens, from which you can choose either to use the same settings used for the document overall, or to enter custom settings. If you choose to use custom settings for this design layer, enter the remaining fields as described for the Document Georeferencing dialog box.

   When your entries are complete, click **OK** to save the settings and return to the Edit Design Layers dialog box.

Click to show/hide the parameters.

Parameter	Description
This layer is georeferenced	
by the document	Specifies that the georeferencing information for this design layer is the same as the settings for the document. Click **Edit Document Georeferencing** to view or change the current document settings on the Document Georeferencing dialog box.
by custom settings	Specifies that the georeferencing information for this design layer is different from the settings for the document (or no document settings were entered). To copy the georeferencing information from an existing layer, click **Copy from Layer** to open the Copy Georeference from Layer dialog box; select a design layer and click **OK**.
Settings fields and Transformation mode	See “Entering Georeferencing Information for the Document” on page 775
Reset	Discards any edits made and returns all fields to their former values

4. Click **OK** to enable georeferencing for the design layer. The selected projection displays in the Organization dialog box.

Entering Georeferencing Information for the Document

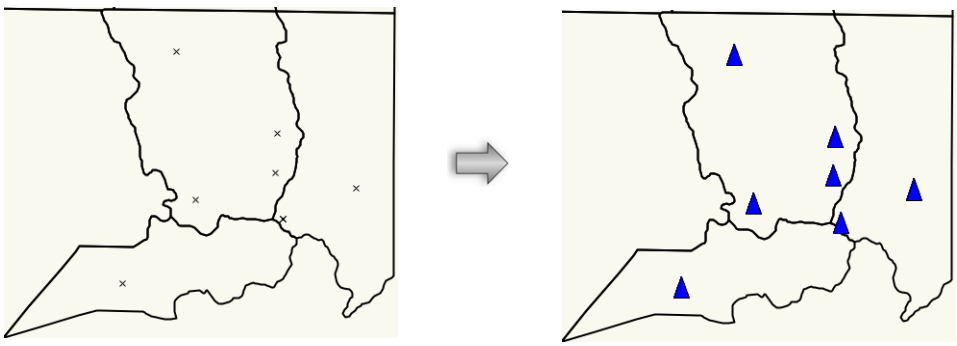
Creating a Graticule
Creating a Great Circle
Projection Options

A L Replacing Objects with Symbols

The **Replace with Symbol** command converts a selected object into a specified symbol. The symbol can use the records provided with the original object, the records associated with the replacement symbol, or both. If an imported Shapefile contains data markers that indicate the location of certain types of objects, you can convert the markers into a symbol that better suits your drawing.

To replace an object with a symbol:

1. Select the object(s) to be replaced.
2. Select **Modify > Convert > Replace with Symbol.**
3. The Choose a Symbol dialog box opens.
4. Select a symbol, and then select whether to use the original object records and the replacement symbol records.
5. Click **OK**, and then confirm that you want to replace all selected objects.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Entering Georeferencing Information for a Design Layer

L Moving an Object to a Specific Location

You can move an object in a georeferenced file to a different geographic location.

To move an object geographically:

1. Select the object(s) to be moved.
2. Select **Modify > Move > Move Geographic**.

   The Move Geographic dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Move to location	Select to move the selected object(s) to the specified **Latitude** and **Longitude**
Latitude/Longitude	Specifies the latitude and longitude to which the object will be moved; enter decimal degrees, or degrees/minutes/seconds (for example, 39.18, 39° 10’ 32”, or 39d 10m 32s) preceded by a minus (-) sign when appropriate
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Move relative	Select to move the selected object(s) from the current location using the specified **Azimuth** and **Distance**
Azimuth/Distance	Specifies the direction in degrees (0° is the Y axis, 90° is the X axis), and the distance the object will be moved, in document units

3. Click **OK** to move the object(s).

Entering Georeferencing Information for a Design Layer

Creating a Graticule

Use the **Graticule** tool to place a reference grid object over the drawing. Unlike a grid that is based on X and Y coordinates, the graticule lines are based on longitude and latitude lines that are inferred from the layer projection.

Mode	Description
Polyline creation options	Selects the method for drawing the polyline that will bound the graticule object; see “Creating Polylines” on page 298.
Fillet Radius	If the Arc Vertex mode is selected, specifies the fillet radius to use
Preferences	Sets the default parameters to be used for the tool

To create a graticule:

1. From the Site Planning tool set, select the **Graticule** tool.
2. From the Tool bar, click the **Preferences** button.

   The Graticule Properties dialog box opens.

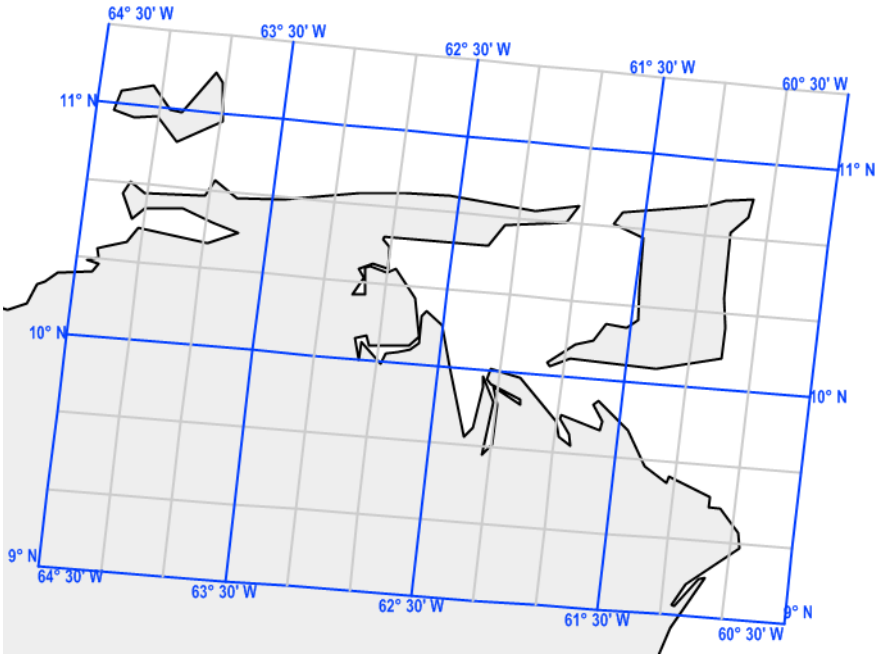
Click to show/hide the parameters.

Parameter	Description
Latitude	
Major Lines Interval	Sets the distance (in degrees) between the major latitude lines
Minor Line Spacing	Sets the number of minor lines to draw between the major latitude lines
Longitude	
Major Lines Interval	Sets the distance (in degrees) between the major longitude lines
Minor Line Spacing	Sets the number of minor lines to draw between the major longitude lines
Label	
Label Major Lines	Places a label on each major latitude and longitude line
Label Minor Lines	Places a label on each minor latitude and longitude line

Parameter	Description
Label Style	Specifies whether the degrees on the labels are in decimal or degrees/minutes format
Attributes	Specifies the pen color and line style for the major and minor lines

3. Click **OK** to save the default graticule settings.
4. From the Tool bar, select the type of polyline to use as the boundary for the graticule.
5. Click on the drawing to create the polyline boundary in the approximate location of the latitude and longitude lines you specified. Make the polyline larger than the graticule will be, so the graticule will not be cut by the polyline. When you complete the polyline, a graticule object is created over the specified location.

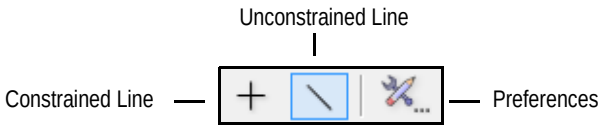
   The Vectorworks software automatically calculates the minimum and maximum values for the latitude and longitude using the geometry and location of the defined polyline. These values can be edited later in the Object Info palette.



Entering Georeferencing Information for a Design Layer

Creating a Great Circle

Use the **Great Circle** tool to place a great circle object over the drawing. Unlike a 2D dimension that shows the linear distance between X and Y coordinates in 2D, a great circle is an arc that shows the shortest distance between two points on the sphere-shaped Earth. Use the **Great Circle** tool (instead of a 2D dimension tool) to measure distances in a georeferenced drawing.



Mode	Description
Constrained Line	Constrains the line to be vertical, horizontal, and 30° or 45° from vertical or horizontal in any direction
Unconstrained Line	Draws the line at any angle Press and hold the Shift key to snap the line to predetermined angles
Preferences	Sets the default parameters to be used for the tool

To create a great circle:

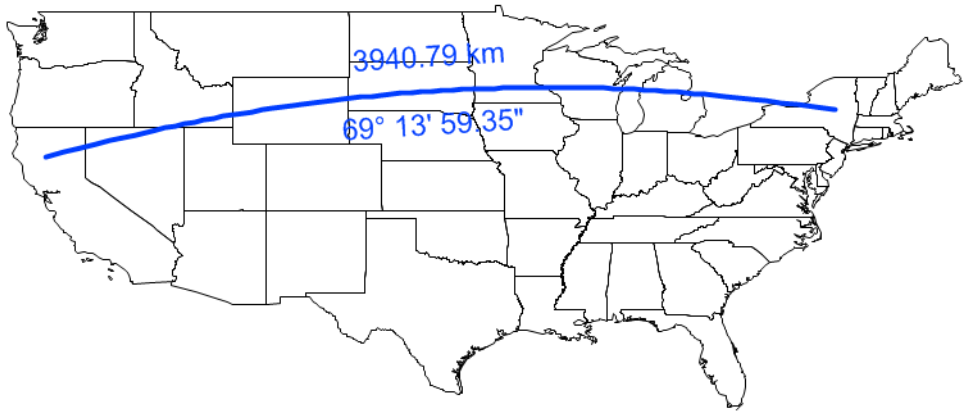
1. From the Site Planning tool set, select the **Great Circle** tool.
2. From the Tool bar, click the **Preferences** button.

   The Great Circle Properties dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Show Distance	Places a distance measurement on the drawing
Distance Unit	Select the unit to use for the distance measurement
Show Angle	Places an angle measurement on the drawing
Angle Format	Select whether to use an azimuth or bearing format for the angle display
Reverse	Reverses the direction of the angle measurement

3. Click **OK** to save the default great circle settings.
4. From the Tool bar, select whether to use the Constrained Line or Unconstrained Line mode when drawing the great circle.
5. Click on the drawing at the beginning point of the great circle, and click again at the ending point to create the object.

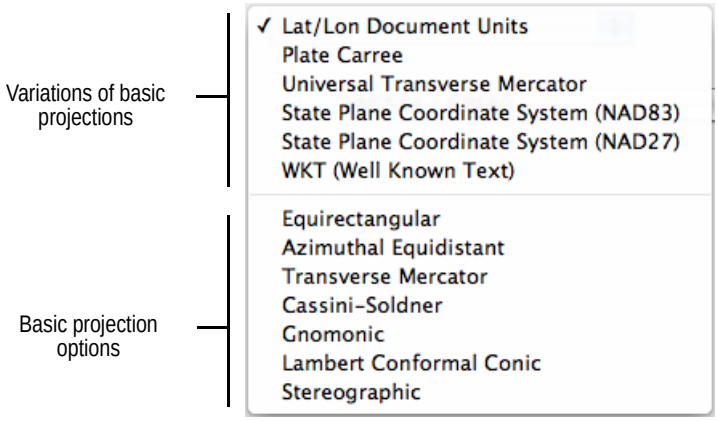


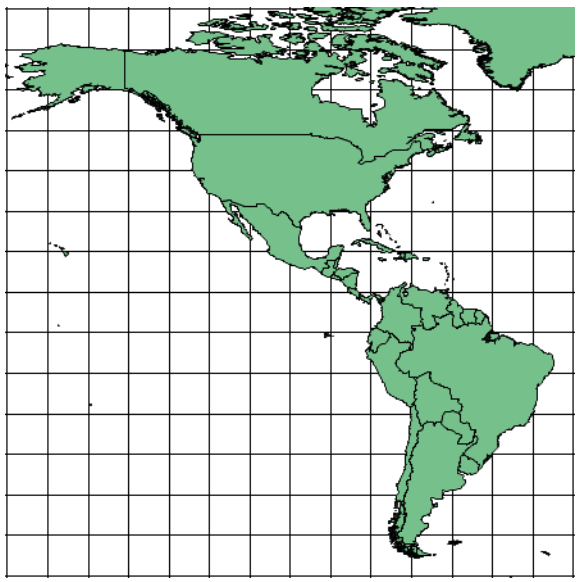
Entering Georeferencing Information for a Design Layer

AL Projection Options

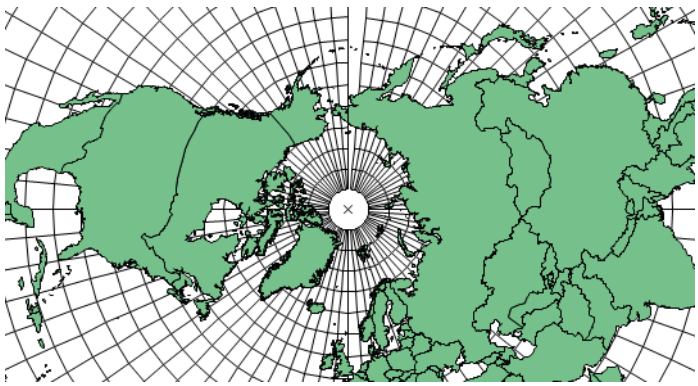
For the Vectorworks Landmark product, there are several options for the type of geographic projection to use for georeferenced design layers. The Vectorworks Architect product has fewer options, because projection is generally less important for drawings on an architectural scale. For larger maps drawn with the Vectorworks Landmark product, however, the proper projection can be very important.

On the Document Georeferencing and the Georeferencing dialog boxes, the projections at the top of the **Projection** list are variations based upon the basic types, which display at the bottom of the list. For example, the Plate Carrée option is based upon the Equirectangular projection. The following table describes the basic projection types.



Projection Type	Description
Equirectangular	Simplest projection, good for maps of the entire world. Latitude and longitude lines are straight and equally spaced. The Lat/Lon Document Units projection is Equirectangular, centered around the equator and prime meridian, and scaled such that one document unit represents one degree of latitude and longitude. The Plate Carrée projection is Equirectangular, centered around the equator and prime meridian, and scaled in real-world units. 

Projection Type	Description
Azimuthal Equidistant	Good for areas centered around a given point, roughly equal in height and width. Straight lines drawn from the center point represent the accurate geographic distance. 
Transverse Mercator	Excellent for mapping narrow (north-south) areas around a chosen longitude. Very great distortion farther east and west. Center longitude defines the chosen meridian; center latitude should be close to the area being mapped. Scale should be 1 or close to it (for example .9996). The Universal Transverse Mercator projection is Transverse Mercator defined by 6° longitude zones. In the US State Plane Coordinate System (SPCS) projections, most US state zones are either Transverse Mercator or Lambert Conformal Conic, depending on the shape of the state and its zones. There are two systems, based on two different North American Datums: NAD27 (now obsolete) and NAD83 (the current standard). 
Cassini-Soldner	Good for mapping narrow (north-south) areas around the central longitude. Distorts shapes farther east and west. 

Projection Type	Description
Gnomonic	Good for navigational maps and small areas around a chosen point. All straight lines represent great circles (shortest geographical distance). 
Lambert Conformal Conic	Good for mapping wide (east-west) areas between chosen parallels. Distortion increases farther from the chosen parallels. In the US State Plane Coordinate System (SPCS) projections, most US state zones are either Transverse Mercator or Lambert Conformal Conic, depending on the shape of the state and its zones. There are two systems, based on two different North American Datums: NAD27 (now obsolete) and NAD83 (the current standard). 
Stereographic	Good for mapping polar areas. Parameters define the center point, which is 90 latitude for the North Pole or -90 latitude for the South Pole. Other points can be used. Scale should be 1 or very close to it (for example .9996). 

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Entering Georeferencing Information for a Design Layer
Entering Georeferencing Information for the Document
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Plants

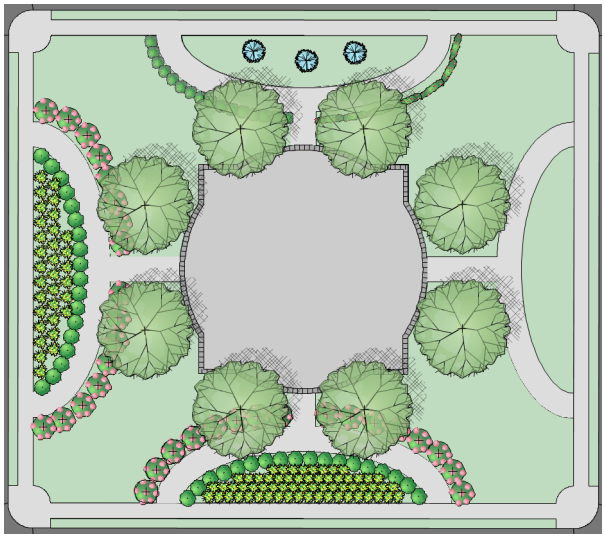
Creating Plant Definitions

The **Plant** tool in the Vectorworks Landmark product both defines and places plant objects. The plant database maintains an extensive set of plant data that can be used in plant definitions.

Placing plants is extremely flexible. Place plants pre-defined with species information and appearance settings, or add generic plants to the drawing and re-assign them to a specific plant species later in the design process. A selected plant can be placed individually or in multiples. Once placed, a plant group can be changed into individual plants, or individual plants can be grouped. In addition, defined and undefined plant masses can be created according to several methods.

A plant definition includes the specific 2D and 3D plant appearance, parameters, and botanical data. Plants are defined by editing a current definition or by creating a duplicate of a current definition. The plant definition is specified by entering parameters or by loading (and editing) botanical data from the plant database.

When the Vectorworks Landmark product is installed, a generic plant objects library is included. In addition, 2D and 3D plant symbols are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219). The resources in the Plant Defaults.vwx file are available in the Plant Settings dialog box.



When plants are defined, the definition parameters apply by default; individual plant settings can be changed before the plant is inserted on the drawing, in the Plant Settings dialog box, or after placement, in the Object Info palette.

Click here for a video tip about this topic (Internet access required).

To create a plant definition:

1. Click the **Plant** tool from the Site Planning tool set.
2. The Plant Settings dialog box opens automatically if plants have not yet been placed in the drawing. Otherwise, click **Preferences** from the Tool bar.

   The Plant Settings dialog box opens.
3. On the Definition pane, select a plant symbol that is close in appearance to the new plant.

   Plants can be placed with just this basic information; see “Adding Plants to the Design” on page 793. The plant definition can be edited later if desired.

4. Click **Edit Definition** from any pane in the Plant Settings dialog box to edit the current definition, or click **Duplicate** from the Definition pane to create a new plant definition based on the selected plant. Alternatively, to edit a plant definition, select a plant from the drawing, select **Edit** from the **Resources** menu or the plant context menu, and select **Definition**.

   Whether creating a new plant from a duplicate, or editing a current plant definition, the Edit Plant Definition dialog box opens, displaying the selected plant symbol. Provide a name for the plant symbol, and then specify the plant parameters on each tab to define the plant. As the parameters are defined, the preview dynamically displays the plant appearance.

   Although the Edit Plant Definition dialog box and the Plant Settings dialog box appear similar, keep in mind that the plant definition parameters define the default plant settings. The default plant settings can be changed at placement, allowing for variations of the same plant definition.

   To automatically add plant information from the plant database, click **Get Plant Data** (see "Using Plant List Data in Vectorworks Landmark" on page 815).

   The plant definition's geometry can be based on a symbol present in the file or from default plant content, from drawing objects in the file, or from another plant; see "Creating Plant Geometry" on page 791.

5. When the default plant parameters have been defined, click **OK** to return to the Plant Settings dialog box. From there, additional plants can be defined, and a plant can be selected for placement.

   When using workgroup referencing, the plant definitions must exist in the same file as the site model they reference, so that their Z values can be set to the site model surface.

   Plant resources can be exported from the Resource Browser or by selecting **Export Plant** from the plant context menu; see "Exporting Custom Resources" on page 234.

Each pane of the plant definition dialog box is described in the following sections.

Plant Definition: Preview

Every pane of the plant definition displays the **Plant Symbol Name** and previews, allows botanical data to be obtained from the plant database, and provides the option to obtain the 2D and 3D plant geometry from a symbol. The previews update as the plant definition changes.

Click to show/hide the parameters.

Parameter	Description
Plant Symbol Name	Enter a name for the plant symbol; this name displays in the Resource Browser
Get Plant Data	Opens the Get Plant Data dialog box; select a plant to add its data to the current plant definition (see "Using Plant List Data in Vectorworks Landmark" on page 815)
Preview	Dynamically previews the plant appearance, showing how it will appear when added to the drawing with the **Plant** tool
Top/Plan Preview	Displays the 2D component of the plant symbol
3D OpenGL Preview	Displays the 3D component of the plant symbol, if one has been added to the symbol definition (image props, such as those provided by xFrog®, can be used as 3D plant geometry)
Copy from Symbol	Obtains the plant geometry from a symbol in the file or from default content; see "Creating Plant Geometry from a Symbol" on page 791

Plant Definition: Insertion Options Pane

Specify the default insertion options for the plant definition on the Insertion Options pane.

If plant data were obtained from the plant database, the plant database values are displayed. Values and settings entered on this tab become the default for the plant definition. These default values can be changed for individual plants.

Click to show/hide the parameters.

Parameter	Description
Spread	Specifies the default plant spread diameter (the maximum width of the mature plant, as drawn) for both single and multiple plant placement
Height	Sets the default height of the typical mature plant at insertion
Spacing	Sets the default spacing between plants at insertion, for the **Poly-Edge Spaced**, **Rectangular Array**, and **Triangular Array** insertion modes

Plant Definition: Schedule Pane

Click the Schedule pane to specify default plant values for display and reporting.

If plant data were obtained from the plant database, the plant database values are displayed. Values and settings entered on this tab become the default for the plant definition. These default values can be changed for individual plants.

Click to show/hide the parameters.

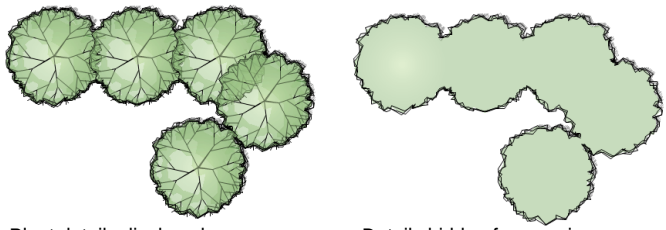
Parameter	Description
Latin Name	Specifies the plant genus and species
Common Name	Specifies the plant common name or names
Plant/TagID	Identifies the plant with a unique code; this code appears in the plant list and on ID tags, if selected (see “Plant ID Codes” on page 799 for the definition of common code categories)
Scheduled Size	Indicates the plant caliper or container size; displays in the Plant List worksheet
Quantity Type	Select the quantity unit for the plant; quantity can be defined as unit count, dripline area, or border area
Price Code (SKU)	Specifies the price code entry in Stock Keeping Units (SKU)
Price	Indicate the plant cost per unit of quantity; the plant cost and quantity are reflected in the Extended Price calculation in the Plant List worksheet
Schedule comments	Specifies default comments about the plant that display in the Plant List worksheet

Plant Definition: Render Pane

Click the Render pane to specify the 2D plant styles. These effects display in Top/Plan view only; plant shadows do not display in worksheet images. A document preference makes it easy to give plans a uniform appearance by applying a document-wide shadow style to all plants and massing models in a drawing (see “Plan Shadows Preferences” on page 63). Even when a document preference is set, plant definitions can be edited on the Render pane to customize the drawing’s appearance. The outline, massing, and shadow parameters can be toggled for all plants in a document by selecting **View > Show > Show or Hide Plant Styles**.

Values and settings entered on this tab become the default for the plant definition. These default values can be changed for individual plants.

Click to show/hide the parameters.

Parameter	Description
Plant Massing	
Mass Outline	Select a plant outline style  The outline's pen attributes can be changed for selected plant objects in the Attributes palette.
Mass Plants	Hides the details of plants that overlap to easily create a plant mass. A continuous outline is created around overlapping plants, and internal details are hidden. The 2D appearance of the massed group is controlled by the back polygon in the plant symbol group, so massing could cause the plant to display without a fill color. (An alternative way of hiding plant details is to use the Plant-Components classes to control the visibility of elements within the plant symbol.) Plants that contain bitmap images cannot be massed. 
Plant Shadow in Plan View	Specifies special shadow effects that display in Top/Plan view only
No Shadow	Select to omit shadows from the Top/Plan view
Use Document Preference Settings	Select to use the settings from the Plan Shadows pane on the Document Preferences dialog box; click **Settings** to review or edit these document-wide shadow settings (see "Plan Shadows Preferences" on page 63)
Custom	Select to create a custom shadow setting for this plant object
Offset	Enter a value for the shadow offset
Offset Units	Select the shadow offset units. Factor of Object Height calculates the offset based on the object's height and the offset value.
Angle	Set the shadow angle by entering a value between -180° to 180° or by using the slider. 0° is straight up.
Fill Style	Select a fill style for the shadow
Color/Resource/ Class	Depending on the fill style selected, select a fill color, resource (hatch, image, gradient, tile), or class for the shadow

Parameter	Description
Opacity	Set the shadow opacity by entering a value between 0 and 100% or by using the slider
Use Class Opacity	Select to use the class's opacity setting (see "Setting Class Properties" on page 179)

Plant Definition: Plant Data Pane

Click the Plant Data pane to view the plant database information for the plant.

To add botanical data from a plant list to the plant definition, click **Get Plant Data** and select a plant (see "Using Plant List Data in Vectorworks Landmark" on page 815).

To change the plant data for the current plant definition only (without changing the plant list data), click on the parameter and click **Edit**. Enter the new value in the Edit Field dialog box and click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Plant Geometry
Adding Plants to the Design
Editing Plants
The Plant Database
Plant Graphics

Creating Plant Geometry

The plant geometry in a plant definition can easily be created from another symbol, from drawing objects, or from another plant.

Creating Plant Geometry from a Symbol

To create plant geometry from another symbol:

1. Begin to create a plant definition as described in "Creating Plant Definitions" on page 787.
2. From the Edit Plant Definition dialog box, click **Copy from Symbol**.

   The Copy From Symbol dialog box opens. Select the plant content file and then the symbol that has the geometry to copy and choose whether to copy the 2D, 3D, or both components of the symbol geometry.

Click to show/hide the parameters.

Parameter	Description
Plant content file list	From the list browser, select the plant content file that contains the symbol to copy; symbols in the selected plant content file are used to populate the Symbol list. The Vectorworks Landmark software is installed with a basic selection of default plant symbols to get you started. Select **Help > Download Content** to download many more plant symbols in the free content libraries provided and/or add your own custom plant symbols (see "Resource Libraries" on page 219). Once added to the library, the appropriate plant content files automatically appear in the list browser. The list browser displays content located in [Vectorworks or User]/Libraries/Objects-Landscape & Site/Plant Imagery 2D+3D; check the filepath if the expected content does not display.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Symbol list	Select the symbol that has the geometry to copy. The list displays the symbols in the current document or selected plant content file; select a different plant content file from the list browser to display additional options.
Explore (Windows) / Reveal in Finder (Mac)	Click to find the selected file in the Windows Explorer / Mac OS Finder
Use only the 2D component of the symbol	Copies only the 2D component of the selected symbol to the plant definition
Use only the 3D component of the symbol	Copies only the 3D component of the selected symbol to the plant definition
Use the 2D and the 3D components of the symbol	Copies both the 2D and 3D components of the selected symbol to the plant definition

3. Click **OK** to copy the selected geometry from the symbol to the current plant definition.

Creating Plant Geometry from Drawing Objects

To create plant geometry from drawing objects:

1. Select the 2D, and optionally, 3D drawing objects to use in a new plant definition.

   A plant symbol can be created from another symbol. The xFrog plant images can be used as a basis for the 2D geometry.

2. Select the **Create New Plant** command from the appropriate menu:

   - Designer workspace: **AEC > Plants > Create New Plant**
   - Landmark workspace: **Landmark > Create New Plant**

   Alternatively, right-click (Windows) or Ctrl-click (Mac) in the drawing, with the geometry or plant symbol selected, and select **Create New Plant** from the document context menu.

   The Edit Plant Definition dialog box opens, with the plant symbol name as “Untitled Plant.”

3. Specify the plant definition parameters, similar to creating a new definition (see “Creating Plant Definitions” on page 787).

4. Click **OK** to create the new plant.

   If no 3D geometry was originally included, the plant symbol contains a 3D locus by default. Once the plant has been created, edit the 3D component of the plant symbol to add the 3D geometry. See “Editing Symbol Definitions” on page 247 for information on editing symbol components. Image props, such as those provided by xFrog, can be used as 3D plant geometry.

Creating Plant Geometry from an Existing Plant

To create plant geometry from an existing plant:

1. Select the existing plant with geometry to convert to a new plant symbol.

2. Select the **Create New Plant** command from the appropriate menu:

   - Designer workspace: **AEC > Plants > Create New Plant**
   - Landmark workspace: **Landmark > Create New Plant**

   Alternatively, select **Create Plant from Object** from the plant context menu, or select the plant to duplicate from the Resource Browser, and select **Resources > Duplicate**, and then **Resources > Edit**.

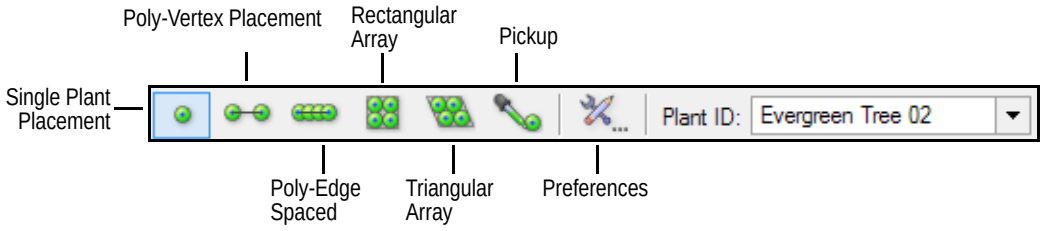
   The Edit Plant Definition dialog box opens, with the plant name symbol as “Untitled Plant.”

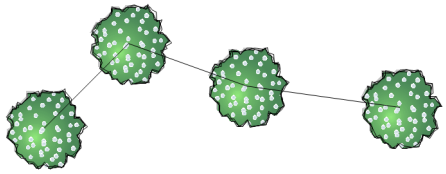
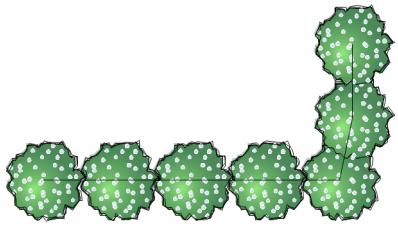
3. The plant graphics and parameters are based on the original plant. Specify new plant definition parameters, similar to creating a new definition (see “Creating Plant Definitions” on page 787).
4. Click **OK** to create the new plant.

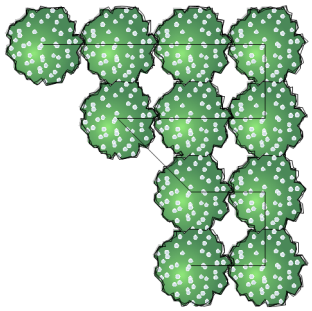
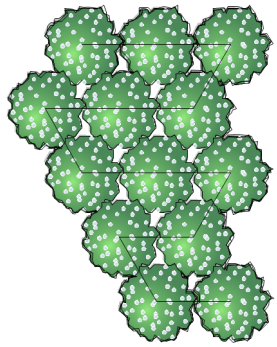
Adding Plants to the Design

Plant placement in the Vectorworks Landmark program is very flexible. Generally, plants are defined first, and then placed; plant parameters are set by the definition, but if necessary, certain parameters can be changed in the Plant Settings dialog box before placement, or in the Object Info palette after placement. This allows variations of the same plant definition, which is an essential part of a landscape designer’s workflow. Another way of placing plants is to place “generic” plants from the default content, and create plant definitions for them later.

In addition to the appearance and parameters defined for plants, the **Plant** tool can place plants in several configurations, from single plants to arrays of plants. When setting parameters after placement, an array of plants is considered to be a single “plant” in the Object Info palette. Plants can also created by drawing a polyline and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).



Mode	Description
Single Plant Placement	Places a single specified plant at each mouse click 
Poly-Vertex Placement	Places plants at each clicked polygon vertex 
Poly-Edge Spaced	Places plants along the drawn polygon at the **Spacing** distance specified in the definition or settings 

Mode	Description
Rectangular Array	Fills the drawn polygon with plants in a rectangular array at the **Spacing** distance specified in the definition or settings 
Triangular Array	Fills the drawn polygon with plants in a triangular array at the **Spacing** distance specified in the definition or settings 
Pickup	Sets the default plant settings to match those of a selected existing plant
Preferences	Opens the Plant Settings dialog box, for specifying the plant to place, its placement options, and its definition
Plant ID	Selects a plant for placement by its ID (**Plant Symbol Name**)

To place plants on the drawing:

1. Click the **Plant** tool from the Site Planning tool set and select the plant to place.

   The Plant Settings dialog box opens automatically if there are no plants in the Resource Browser. Otherwise, click **Preferences** from the Tool bar to select the plant to place, and specify the plant settings if they are different from the definition. The plant settings are described in the following sections.

   The plant to place can also be selected by its plant symbol name from the **Plant ID** list on the Tool bar. The plant resources in the active drawing display alphabetically at the top of the list; the remaining plants are from the default content resources.

   Double-click a plant from the Resource Browser to activate the **Plant** tool and place the selected plant.

   The **Plant** tool uses these settings until they are changed again by selecting a different plant ID from the Tool bar, clicking **Preferences** from the Tool bar, or until **Pickup** mode is selected, which changes the default settings to those of a selected existing plant.

2. Select the plant placement mode from the Tool bar.

3. Depending on the placement method selected, either click in the drawing to place a single plant, or draw a polygon. As the **Plant** tool is clicked in the drawing, a preview of the plant spread is displayed to help with plant placement.

   The **Plant** tool parameters are retained so that the successive placement of plants is easily accomplished.

Each pane of the plant settings is described in the following sections. As the parameters are defined, the preview dynamically displays the plant appearance.

Do not be confused by the similar appearance of the Plant Settings dialog box and the Edit Plant Definition dialog box; keep in mind that the plant definition parameters define the default plant settings. The default plant settings can be changed at placement if needed, allowing for variations of the same plant definition. Default plant settings that do not require editing can be left as defined by the plant definition.

Plant Settings: Definition Pane

The Definition pane can be used to select the plant definition for placement by the **Plant** tool. From any pane, you can also edit the plant definition parameters or create a duplicate of the plant definition, as described in “Creating Plant Definitions” on page 787.

Click to show/hide the parameters.

Parameter	Description
Definition	Select the plant definition to insert from the graphical plant symbol list. The list displays the symbols in the current document or selected plant content file; select a different plant content file from the list browser to display additional options.
Latin Name/Common Name	Displays the selected plant’s definition information
Duplicate	Click to create a duplicate plant based on the selected plant’s definition; see “Creating Plant Definitions” on page 787
Plant Content File list	From the list browser, select the plant content file that contains the plant definition to insert; plant definitions in the selected plant content file are used to populate the **Definition** list. The Vectorworks Landmark software is installed with a basic selection of default plant definitions to get you started. Select **Help > Download Content** to download many more plant definitions in the free content libraries provided and/or add your own custom plant content (see “Resource Libraries” on page 219). Once added to the library, the appropriate plant content files automatically appear in the list browser. The list browser displays content located in [Vectorworks or User]/Libraries/Objects-Landscape & Site/Plant Tool-Symbol+Definition Library; check the filepath if the expected content does not display.

Plant Settings: Insertions Options Pane

The Insertion Options pane displays the default plant insertion settings from the plant definition. To override the plant definition parameters, select the parameter check box and enter the custom value. In addition, if the plant should be included in plant report worksheets, select **On Plant List**.

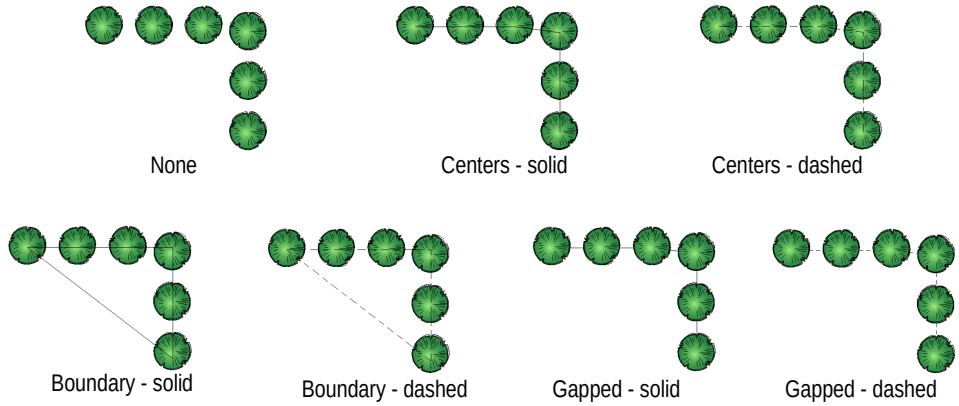
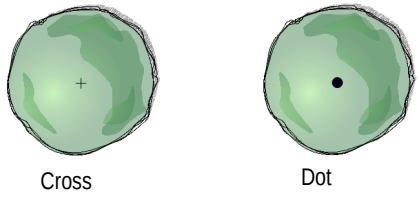
Click to show/hide the parameters.

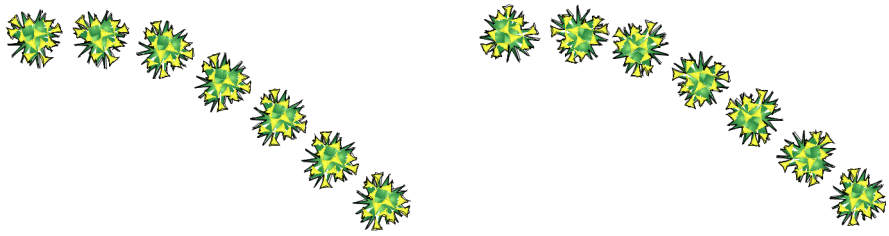
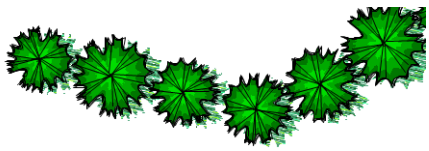
Parameter	Description
Custom Spread	Specifies the plant spread diameter (the maximum width of the mature plant, as drawn) for both single and multiple plant placement
Custom Height	Sets the height of the typical mature plant at insertion
Custom Spacing	Sets the spacing between plants at insertion, for the for the **Poly-Edge Spaced**, **Rectangular Array**, and **Triangular Array** insertion modes
On Plant List	Includes the plant in plant report worksheets

Plant Settings: Annotation Pane

Select plant display parameters on the Annotation pane.

Click to show/hide the parameters.

Parameter	Description
Polygon Display	For multiple plant placements, changes the display of the boundary or center polygon defining the plant cluster shape. 
Tick Style	Select a tick mark (plant center mark) style 
Tick Size	Specifies the tick mark size

Parameter	Description
Plant Rotation	For multi-plant placements, orients the plants in the horizontal direction, rotates the plants along the line created by the drawn polygon, or rotates the plants randomly, creating a more natural appearance  Plants rotated along the line Plants randomly rotated along the line
Plant Scale	Keeps the plants at a fixed size, or randomly varies the plant scale according to the specified percentage of variation  Plants randomly rotated and scaled

Plant Settings: Render Pane

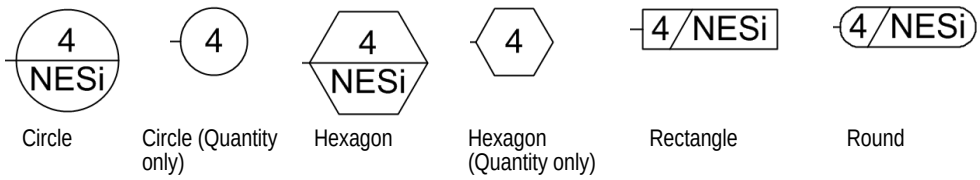
The Render pane displays the default render settings from the plant definition. The **Use Plant Definition** settings indicate that the plant definition render settings apply; see “Plant Definition: Render Pane” on page 789. To override the plant definition parameters, select the **Custom** outline and massing option, and/or select the **No Shadow**, **Use Document Preference Settings**, or **Custom** shadow setting.

Plant Settings: Tag Pane

Set the display of the plant ID tags on the Tag pane. To create a custom plant tag, see “Creating a Custom Plant Tag” on page 798. After creation, the plant tag’s appearance can be modified via the Plant Settings Tag pane, Object Info palette, tag class settings, and the control point locations on the drawing (see “Plant Tag Appearance” on page 800).

Click to show/hide the parameters.

Parameter	Description
Display	Select On to display a plant tag to the right or left of the leader line; set the left or right position with the tag **Shoulder Angle**.
Class	Select a class for the plant tags, to control the tag appearance (line and marker style) and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, or select the default class name.
Approach Angle	Specifies the angle of the leader line, from 0 to 360°. Setting the same angle for several selected plants aligns their leader lines, for an attractive planting plan (see “Aligning and Distributing Leader Lines” on page 1035).

Parameter	Description
Shoulder Angle	When a tag shoulder line is enabled, sets the angle of the shoulder line and, if selected, tag display, from 0 to 360°. To display the plant tag to the left, specify an angle greater than 90 or less than 270 degrees. To display the plant tag to the right, specify an angle less than 90 or greater than 270 degrees. Setting the same angle for several selected plants aligns their shoulder lines, for an attractive planting plan.
Bubble	Specifies the bubble style, if any, for displaying quantity or quantity and Plant ID with the plant tag  Circle; Circle (Quantity only); Hexagon; Hexagon (Quantity only); Rectangle; Round
Top / Center / Bottom	The custom tags present in the drawing and the predefined plant record field combinations for tags are listed. Specify the information to display on each level of the plant tag, or select **None** to hide a level of the plant tag. For the center level, select **Continuation Line** to continue the shoulder as a dividing line between the top and bottom tag information. Alternatively, select **Set Custom Tag** to define a custom plant tag (see "Creating a Custom Plant Tag" on page 798).
Enable Tag Shoulder Line	Adds a shoulder line to the leader line; adjust the shoulder angle with the shoulder control point, or by entering a tag **Shoulder Angle**
Snap Tag to Plant Centers	Snaps the end of the leader line to the center of the plant; deselect to adjust the endpoint location manually
Display Tag Line Marker	Displays a marker at the end of the leader line; specify the marker to use by editing the plant tag class (see "Setting Class Properties" on page 179)

Plant Settings: Schedule Pane

The Schedule pane displays the default schedule settings from the plant definition; see "Plant Definition: Schedule Pane" on page 789. To override the plant definition parameters, select the parameter check box and enter the custom value.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Plants
Creating Plant Definitions
Creating Plant Geometry
The Plant Database
Plant Graphics
Converting Plant Clusters

L Creating a Custom Plant Tag

In addition to providing several predefined plant tags, Vectorworks Landmark allows designers to create custom plant tags.

1. From the Tag pane of the Plant Settings dialog box, select **Set Custom Tag** from one of the **Top / Center / Bottom** fields. Settings for the plant tag currently displaying in this field are displayed in the Set Custom Tag dialog box.

   The Set Custom Tag dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

If editing an existing tag through the Object Info palette, the selected tag's data displays.

2. Select values from predefined plant record fields in the order they should be listed, and type the delimiter text to place between the fields. Select <New Line> to start the next tag item on a new line. Up to six different combinations of fields, delimiters, and new line separators can be used. Entering a custom tag in the **Top** field of the Plant Settings dialog box will insert the custom tag above the reference line, a custom tag in the **Center** field will replace the reference line, entering a custom tag in the **Bottom** field will appear below the reference line.

   The **Tag Appearance** field displays a static text preview of the custom tag. To preview a tag in its entirety, click-drag the bottom right corner of the dialog box to resize it.

3. Click **OK**.
4. Repeat steps 1 through 3 for the other **Top / Center / Bottom** fields on the Plant Settings dialog box, as appropriate, to complete creation of the custom tag.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Plant Settings: Tag Pane
Plant Tag Appearance
Adding Plants to the Design

Plant ID Codes

The meaning of commonly used plant ID code categories is provided.

Plant ID Code	Meaning
A-#	Annual #
CTD	Conifer Tree Display
CTG	Conifer Tree Generic
ETD	Evergreen Tree Display
ETG	Evergreen Tree Generic
G-#	Grasses #
OT-M	Ornamental Tree Massed
OTD	Ornamental Tree Display
OTF	Ornamental Tree Flowering
OTG	Ornamental Tree Generic
OTM	Ornamental Tree Multi-Stem
OTP	Ornamental Tree Patio
P-1 — P-9	Perennials
P1 — P4	Palms
SD-#	Shrub Display #
SD#	Shrub Deciduous #
SDM	Shrub Display Massed
SG#	Shrub Evergreen #
SG	Shrub Generic
SN#	Shrub Needle #
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Plant ID Code	Meaning
STG	Shade Tree Generic
STL	Shade Tree Large
STM	Shade Tree Massed
STP	Shade Tree Patio
STS	Shade Tree Street

Editing Plants

Plant Properties

Plant properties are displayed, and can be edited, in the Object Info palette. As plants are placed, they take on the properties of the associated plant definition and/or the properties in the Plant Settings dialog box. Changes made in the Object Info palette for the selected plants apply to those individual plants only; changes are not reflected in the plant definition.

For multi-plant placement options, edit the polygon defining the plant with the **Reshape** tool if necessary; the plant placement automatically adjusts to fit the new shape.

Click to show/hide the parameters.

Parameter	Description
Plant Settings	Opens the Plant Settings dialog box, for viewing and editing plant parameters
Definition	Displays the plant definition parameters and allows the spacing to be edited as described in “Plant Settings: Insertions Options Pane” on page 795
Replace Plant	Opens the Replace Plant dialog box; select the plant to replace the currently selected plant or all plant instances. (Alternatively, select **Replace Plant** from the plant context menu.)
Annotation/ Render	Edits the plant annotation parameters as described in “Plant Settings: Annotation Pane” on page 796 and the render parameters as described in “Plant Definition: Render Pane” on page 789
Tag	Edits the plant tag parameters as described in “Plant Settings: Tag Pane” on page 797
Vertex parameters	Edits the plant vertices for multi-plant placements; see “Editing Vertex-Based Objects” on page 1002

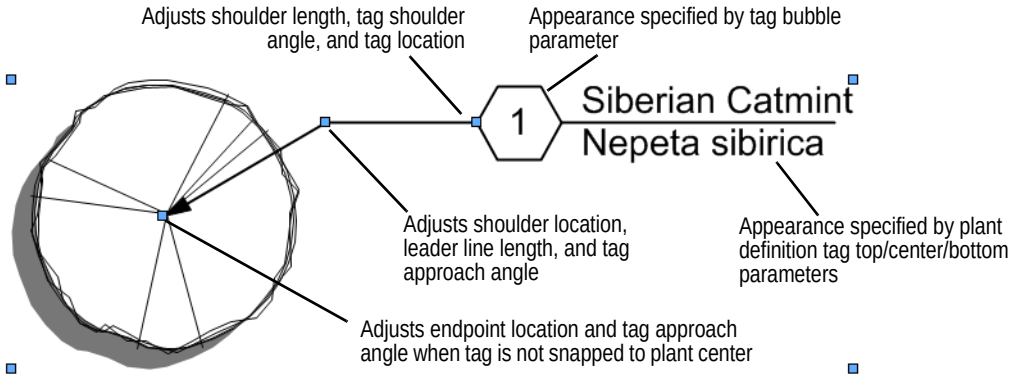
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Plant Tag Appearance
Editing Plant Attributes
Replacing Plants
Modifying Plant Clusters

Plant Tag Appearance

Plant tag appearance and placement is flexible, and can be adjusted in several ways, including through the plant settings, Object Info palette, tag class settings, and the control point locations on the drawing.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Individual plant tags can then be changed for selected plants by adjusting the plant tag parameters in the Object Info palette. For example, selected plants in one area of a planting plan look more uniform when they all use the same tag approach and tag shoulder angle.
- The plant tag class controls the appearance of the leader/shoulder lines, as well as the marker style.
- To move the plant tags of several selected plants at once, click the Unrestricted Interactive Scaling mode of the **Selection** tool.
- To align plant tags for improved readability, use the **Align/Distribute Leader Lines** command (see “Aligning and Distributing Leader Lines” on page 1035).
- If an individual tag needs to be repositioned, plant tags also have several control points for adjusting the tag text and leader line and shoulder position and angle.



Plant Settings: Tag Pane

Editing Plant Attributes

Plants are hybrid symbols, containing a 2D symbol, and optionally, a 3D symbol. As plant definitions are created, the plant symbol is automatically imported into the current file and appears in the Resource Browser. Plants are “red” plug-in object symbols (see “Symbol Types” on page 237 for information on symbol types). The 2D and 3D plant graphics are scaled by the plant definition height and spread parameters.

Because plants are red symbols, plant attributes cannot be directly set or modified from the Attributes palette. Instead, edit the plant symbol components.

To edit plant symbol attributes:

1. Select the plant symbol in the Resource Browser. Select **Resources > Edit**, or select **Edit** from the Resource Browser context menu.

   Alternatively, select a plant in the drawing window, and double-click or select **Edit** from the context menu.

   The Edit Plant dialog box opens.

Click to show/hide the parameters.

Parameter	Description
2D/3D Graphics	Edits the 2D or 3D symbol components (geometry, or shape, and attributes like color and line weight) as described in “Editing Symbol Definitions” on page 247
Definition	Edits the definition of the plant symbol as described in “Creating Plant Definitions” on page 787)

Parameter	Description
Path	For multiple plant placements, selects the boundary or center polygon defining the plant cluster shape, and automatically activates the **Reshape** tool for editing the path

2. Edits to the 2D or 3D components immediately affect all instances of the symbol. Changes to the plant definition affect all future instances of the plant.

Replacing Plants

When replacing plants, select whether to replace the current plant only, or all instances of the selected plant.

To replace plants:

1. Select the plant(s) to replace, or select a plant that is a representative of the plant instances to replace.
2. From the Object Info palette, click **Replace Plant**.

   Alternatively, Right-click (Windows) or Ctrl-click (Mac) on the plant and select **Replace Plant** from the context menu.

   The Replace Plant dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Graphical plant list	Select the replacement plant from the list of plant symbols in the file and in default content
Selected Plants Only	Replaces the selected plant(s) with the new plant
All Instances	Replaces all instances of the plant with the new plant. This does not affect the plant definition of the replaced plant, but simply replaces all plants of one type with another.

3. Click **OK** to replace the plant(s) or all the plant instances with the new plant.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Plants to the Design
Creating Plant Definitions

Modifying Plant Clusters

Plants placed in a multiple placement mode of the **Plant** tool are associated as a cluster. The cluster moves together, and parameter changes affect all plants in the cluster. However, it is possible to dissociate the cluster to make individual plant changes. In addition, new clusters can be created with different combinations of plants. Clustering identical plants which are in close proximity can be desirable for labeling and identification purposes.

Converting Plant Clusters

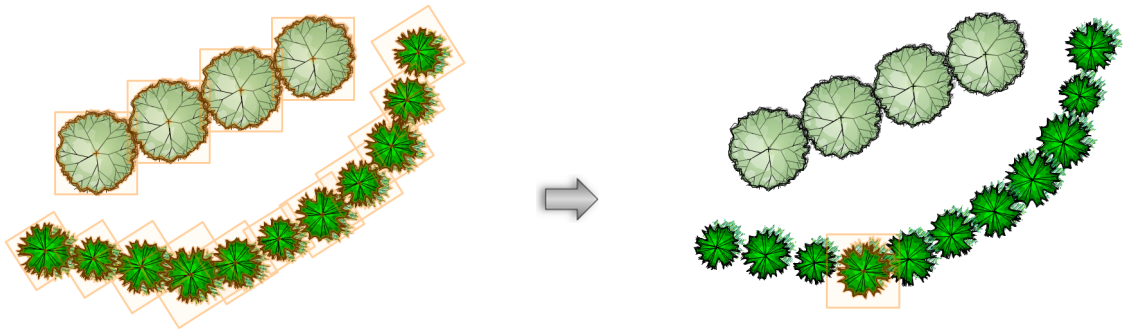
One or more plant clusters can be disassociated to change individual plant parameters or location.

To convert a plant cluster to individual plants:

1. Select the plant cluster or clusters to convert.
2. Select the **Change Plant Grouping** command from the appropriate menu:
   - Designer workspace: **AEC > Plants > Change Plant Grouping**
   - Landmark workspace: **Landmark > Change Plant Grouping**
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Alternatively, right-click (Windows) or Ctrl-click (Mac) on the plant cluster and select **Change Plant Grouping** from the context menu.

3. If one plant cluster is selected, the plants in the cluster are automatically converted to individual plants.
4. If more than one plant cluster is selected, the Choose Mode dialog box opens.
5. Select **Convert Selection into Individual Plants**.
6. Click **OK**. The plants are converted, retaining their original plant type; they can be moved and changed individually.



Combining Plants

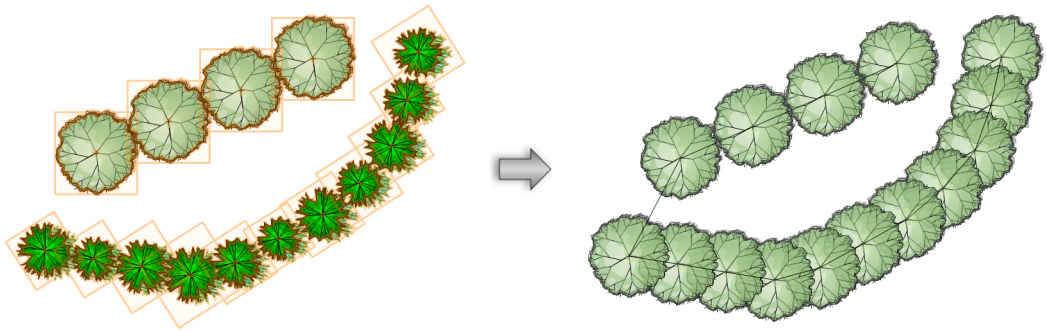
Individual plants and plant clusters can be combined into a single plant cluster. The converted cluster will be a multi-plant placement at polygon corners.

To convert plants to a plant cluster:

1. Select the individual plants, plant clusters, or combination of individual plants and clusters to convert.
2. Select the **Change Plant Grouping** command from the appropriate menu:
   - Designer workspace: **AEC > Plants > Change Plant Grouping**
   - Landmark workspace: **Landmark > Change Plant Grouping**

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the plant and select **Change Plant Grouping** from the context menu.
3. If the selection consists of individual plants of the same type, they are automatically converted to a single plant cluster.
4. If individual and clustered plants are selected, the Choose Mode dialog box opens.
5. Select **Combine Plants into One Single Plant**.
6. Individual and clustered plants of the same type are automatically converted to a single plant cluster.

   If the selection consists of more than one plant type, the Please Choose Plant dialog box opens.
7. All the plants in the selection will be converted to one of the plant types. Select the plant type from the list and its **Plant Spread** value is displayed. Click **OK**.
8. The selected plants are converted to a single cluster of identical plant types.



Plants are clustered based on their drawing order. If the joining polygon of the resulting plant is not as expected, change the drawing order of the plants prior to joining them.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Plant Attributes
Replacing Plants
Creating Plant Definitions

The Plant Database

The plant database provided with the Vectorworks Landmark product manages an extensive list of plant names and botanical information which can be used to associate specific plant data with plant definitions.

The plant database is a stand-alone FileMaker® application that opens in a separate window. The benefits of using the FileMaker database for plant data management include improved navigation, editing, searching, and filtering, as well as the ability to import data from industry-standard sources. At installation, a large set of plant records is provided.

FileMaker is a comprehensive data-management program. The details of every menu command and option are beyond the scope of this manual. However, all relevant information for using FileMaker in conjunction with the Vectorworks Landmark product is presented. Familiarity with FileMaker is not required to use the database or to manage botanical data, and to use that data in the Vectorworks program. The full version of FileMaker can also be used when working with the plant database. For in-depth information about FileMaker, including product documentation, visit www.filemaker.com and navigate to the Support area. (The plant database may not contain all the functionality documented for the full version of FileMaker.)

The general workflow consists of managing the botanical information in the plant database application, and from there, creating the plant lists to be used in the Vectorworks Landmark product.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Accessing the Plant Database
Importing Plant Database Information
Viewing Plant Database Records
Searching for Plants
Managing Plant Database Records
Accessing Plant Information from the Internet
Creating Plant Lists for Vectorworks Landmark
Using Plant List Data in Vectorworks Landmark
Plant Database Field Mapping

Accessing the Plant Database

Accessing the plant database for the first time requires initial setup.

To access the plant database for the first time:

1. Select the **VW Plants Database** command from the appropriate menu:
   - Designer workspace: **AEC > Plants > VW Plants Database**
   - Landmark workspace: **Landmark > VW Plants Database**

   Alternatively, right-click (Windows) or Ctrl-click (Mac) in the drawing and select **VW Plants Database** from the context menu.

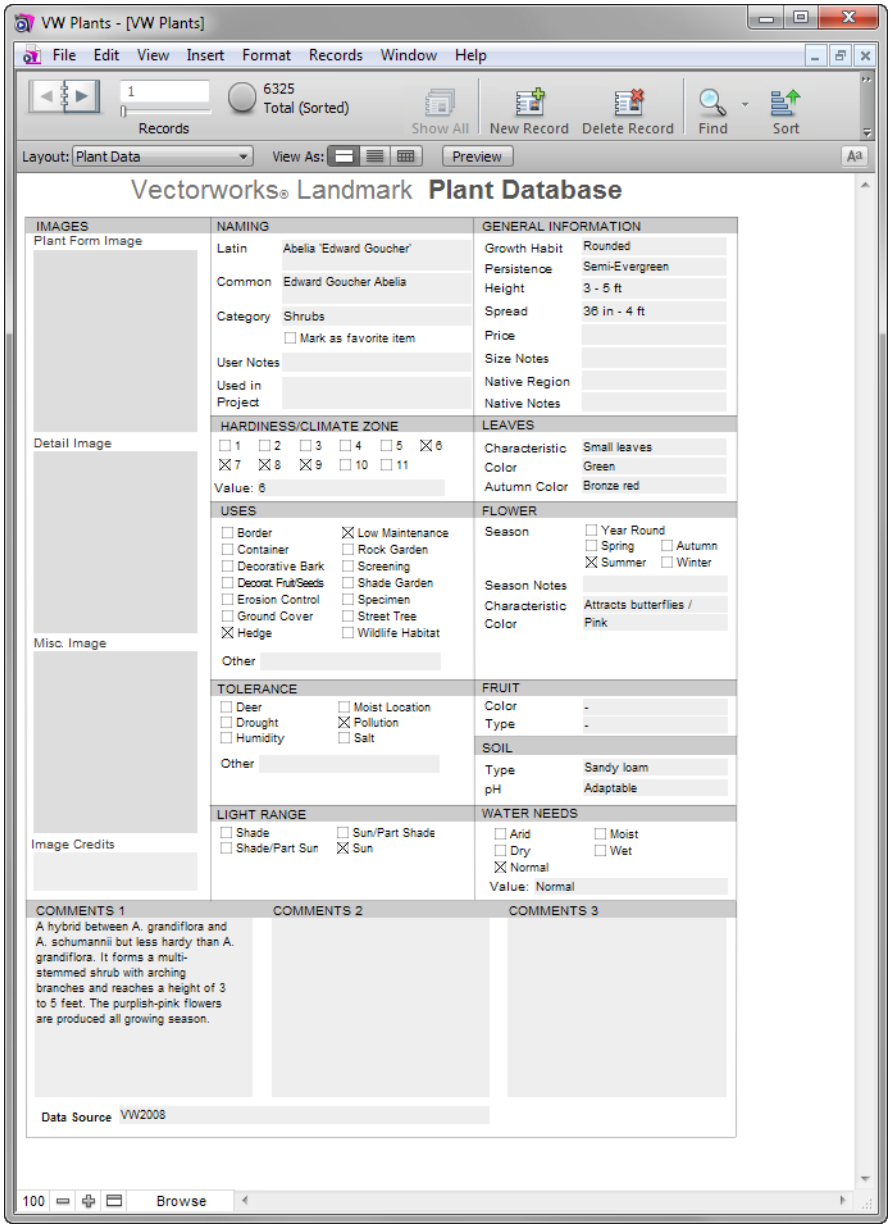
   The Choose Plant Database Location dialog box opens.
2. Indicate where the plant database information is to be located. By default, the plant database is stored locally, in the user folder of the computer where the Vectorworks software is installed. Larger offices that wish to share the plant database among several computers can store and access the plant database from a workgroup folder located on a central computer or server. Alternatively, select **Browse for database folder** and then click **Browse** to specify the desired location.

   If this is the first time the plant database is being accessed and it does not yet exist in the specified location, the database is automatically created in that location. Whenever the plant database is opened in the future, it will use the data from that location. The location can be changed at any point by selecting the **Choose VW Plants** command from the appropriate menu:
   - Designer workspace: **AEC > Plants > Choose VW Plants**
   - Landmark workspace: **Landmark > Choose VW Plants**

   It is possible to maintain several plant databases and switch among them with this command.
3. Click **OK**. The creation of a new plant database location may take a few moments.

   The VW Plants dialog box may open. Specify a user name for the FileMaker application.
4. Click **OK**.

   The Vectorworks Landmark plant database opens, in a separate window.



Once the plant database has been set up, it is accessed immediately when selecting **Landmark > VW Plants Database**. The plant database can also be accessed when creating a plant definition, by clicking **Open VW Plants** from the Get Plant Data dialog box.

The Plant Database

Importing Plant Database Information

A default set of plant database records is provided at installation. The plant database supports several data formats for import, including tab-delimited files, Excel® spreadsheets, .xml files, and many others. The import folder feature allows movies and images to be imported. In addition, plant lists from previous versions of the Vectorworks program can be imported.

To import plant database information:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.

2. Select the **File > Import Records** database command. Files, folders, or .xml files can be imported. Select this command to import plant lists from Vectorworks program versions prior to 12.

   To import plant data from Vectorworks versions 2008 through 2012, first select the **File > Convert VW Plants** database command to convert the plant data to the most recent FileMaker format. Then, import the data with **Import Records**.

   To import plant lists from Vectorworks 12.x, select the **File > Import Vectorworks 12 Plant List** database command.

3. After specifying the location of the file(s) to import, field mapping is required to import the data correctly into the current database. (For Vectorworks 12.x plant lists, field mapping is automatically performed.)

   The important mapping fields for proper use in the Vectorworks Landmark product are described in "Plant Database Field Mapping" on page 817. If more information is required, consult the Support area of www.filemaker.com.

   Click here for a video tip about this topic (Internet access required).

Adding Additional Plant Data

Additional plant datasets may be located in the Plant Database\VW Plants\Additional Datasets folder. These files typically contain regional plant sets provided by a distributor or other plant sets installed with the Vectorworks Landmark software. The field matching has already been performed for these data sets, and they are easy to add to the plant database.

To add plant data from the Additional Datasets folder:

1. Open the plant database as described in "Accessing the Plant Database" on page 804.
2. Select the **File > Add Additional Plant Data** database command.

   Select the file to import. Field mapping is automatic, and the plant information is added to the plant database.

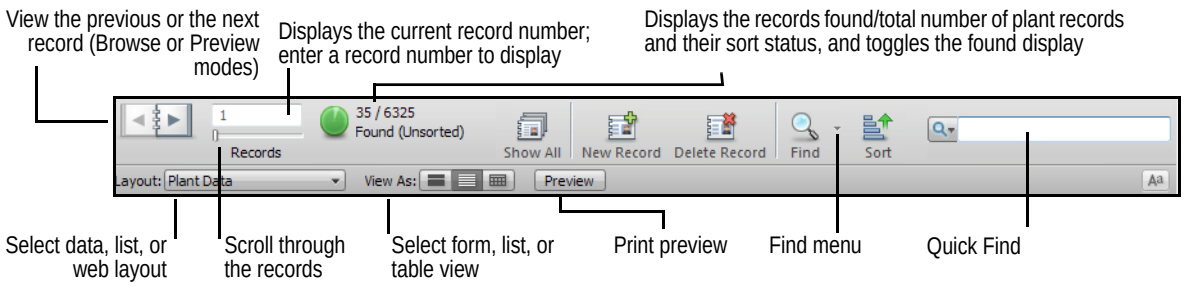
The Plant Database

Viewing Plant Database Records

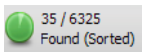
Each plant entry and its associated botanical information is considered a "record" in FileMaker. Several view modes display the records in ways designed to facilitate performing related tasks. When the plant database first opens, the records operate in Browse mode, with Forms displayed.

To familiarize yourself with the different display modes, and view the plant records in a variety of ways:

1. Open the plant database as described in "Accessing the Plant Database" on page 804.
2. Select the **View > Status Toolbar** database command to enable viewing controls and status at the top of the record form. Many of these items can also be found on the **View** and **Records** menu.



3. From the Status toolbar and the **View** menu, select the view mode and layout options depending on the task to accomplish.

Mode or View Option	Description
View Mode	
Browse	Displays plant record information and allows editing
Find	Displays a blank form for specifying search criteria
Preview	Displays plant record information as it will be printed
View as Form	Displays each record as an individual form or “page”
View as List	Displays records consecutively in a scrollable list
View as Table	Displays each record as an item in a table, for sorting and reordering
Layout	
Data View	Displays plant record information
List View	Displays limited plant data for each plant, for sorting and reordering
Web View	Displays a special browser (using the default system browser) for locating plant images and information
Records Found toggle and status 	After a search, the green toggle button displays an approximate pie chart and the number of records found; click the green pie chart button to toggle the display to records that are not in the found set. The sort status is also indicated.
Show All	Clears the results from a search, displaying all records
New Record	Adds a plant record to the database
Delete Record	Deletes the currently selected record from the database
Find	Switches to Find mode layout, for searching
Find menu	Allows searches to be saved, modified, recalled, and cleared
Sort	Opens the Sort Records dialog box, for determining the sort order for the plant records, or for unsorting the records
Quick Find	Searches for the text entered or recent searches, without switching to Find mode
Preview	Displays the records in a Print Preview mode

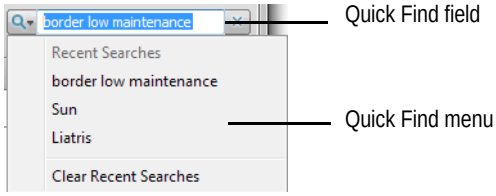
The Plant Database

Searching for Plants

The plant database can be searched from Quick Find in the Status toolbar or by switching the view to Find mode. Quickly search by entering text or repeating a recent search, or conduct flexible, sophisticated searches by combining search criteria. Search criteria can be saved and managed, making it easy to re-create plant lists.

Quick Find Search

Quick find searches the plant records for text.



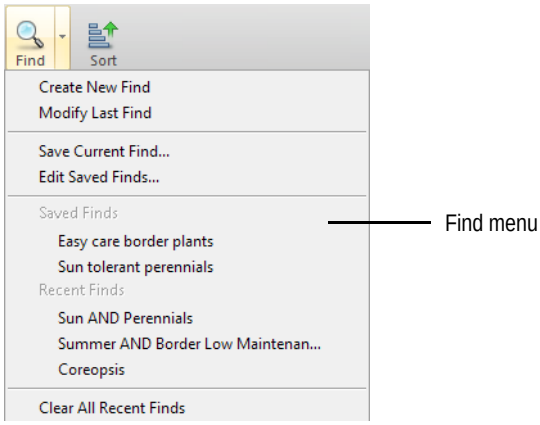
To perform a quick find search:

1. Type one or more search terms into the Quick Find field in the Status toolbar.
2. Press Enter.
3. The records that contain the search terms are displayed.

   To quickly repeat a previous search, click on the **Quick Find** menu and select the search term used previously.

Find Mode

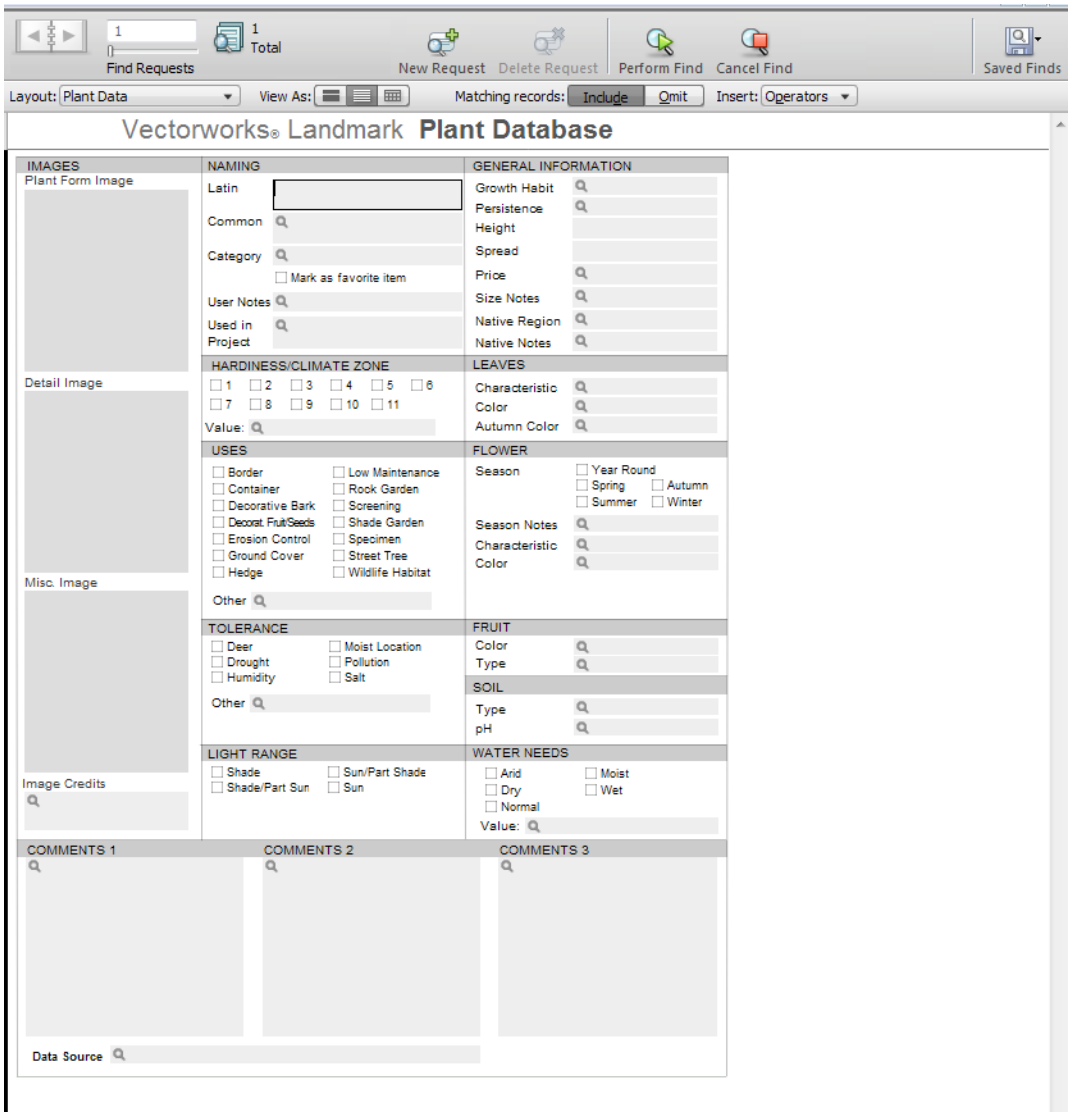
Find mode performs a more advanced search based on specific criteria. Searches can be saved and managed.



To search for plants in Find mode and manage found sets:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Select the **View > Find Mode** database command or click **Find** from the Status toolbar. Alternatively, select **Find > Create New Find**.

   A blank form opens and the display switches to Find mode. Find mode functions are available from the Status toolbar.



3. Enter the criteria for searching.

   Examples include searching for the Latin name “Liatris,” Light Range “Sun,” with a Height > 3-5 ft.

   Click the **Omit** button in the Status toolbar (or select the Omit options from the **Records** menu) to exclude, rather than search for, the criteria. Use the Operators list for even more specific searches.

4. Click **Perform Find** from the Status toolbar, or press Enter.
5. The view mode automatically switches to Browse, and the records that meet the search criteria are displayed.
6. To save the current set of search criteria, select **Find > Save Current Find**.

   Other search criteria set maintenance can be performed from the **Find** menu, to modify, edit, clear, and select saved searches. A previously saved search is executed by selecting its name from the **Find** menu.

   To return to viewing all the plant records, select the **Records > Show All Records** database command or click **Show All** from the Status toolbar.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Plant Database
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Managing Plant Database Records

Editing Plant Records

Plant database records can be edited in Browse mode, whether in form, list, or table view (see “Viewing Plant Database Records” on page 807).

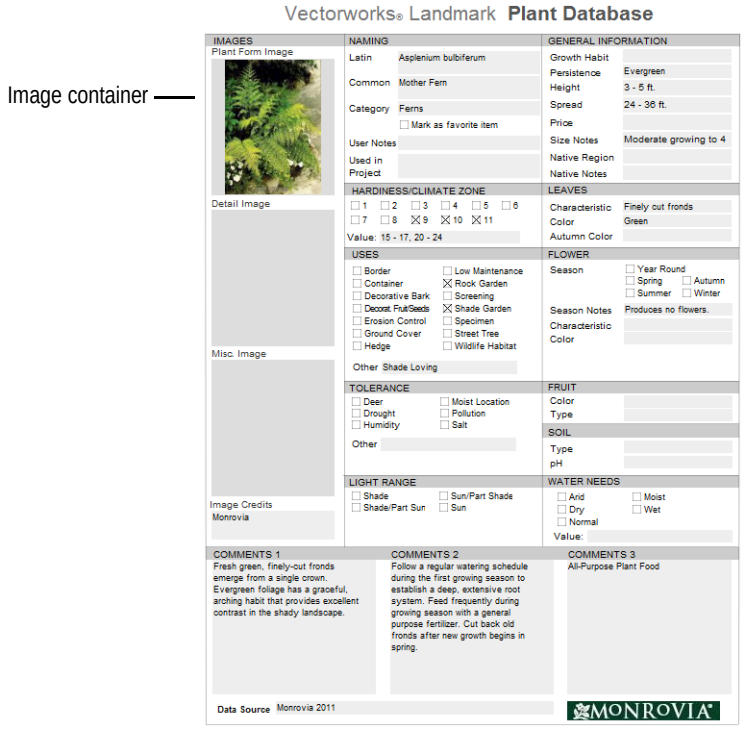
To edit plant records:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Select the **View > Browse Mode** database command.
3. Select the plant record to edit by scrolling or searching.
4. Changes made to the fields and check boxes are automatically saved.

   Useful edits include:

   - Marking a plant as a Favorite (to search for favorite plants later)
   - Indicating the Project Information (to track plants by project)
   - Adding custom information to drop-down lists
   - Placing an image, video, audio, pasted text, or link to an embedded object into one of three containers; right-click on an image container to access the options
   - Adding image credits and data source information to avoid copyright issues
   - Adding extra plant or project information that is useful in the Vectorworks program

   Switch between metric and imperial units for the Height and Spread fields with the **Edit > Options > Use Imperial Value Lists** or **Use Metric Value Lists** database commands.



Project ID numbers and favorites can be easily be added to all plants found by a search, with the **Records > Add Project ID** and **Records > Mark As Favorite** database commands. These fields export to the Vectorworks program and are useful for tracking and finding plants. If the Project ID is no longer needed at the completion of a project, search for all plants with that ID and then select the **Records > Delete Project ID** database command to remove it.

Adding Plant Records

To add a plant to the plant database list:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Select the **View > Browse Mode** database command.
3. Select the **Records > New Record** database command or click **New Record** from the Status toolbar.

   New records are appended to the end of the record set.
4. Enter the plant information. Information is saved automatically.

Deleting Plant Records

To remove a plant from the plant database list:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Locate the record or records to delete, by searching or scrolling.
3. Select the **Records > Delete Record** database command or click **Delete Record** from the Status toolbar to delete an individual record, or **Records > Delete Found Records** to delete a found set of records.

   Confirm the deletion; this action cannot be undone.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Plant Database

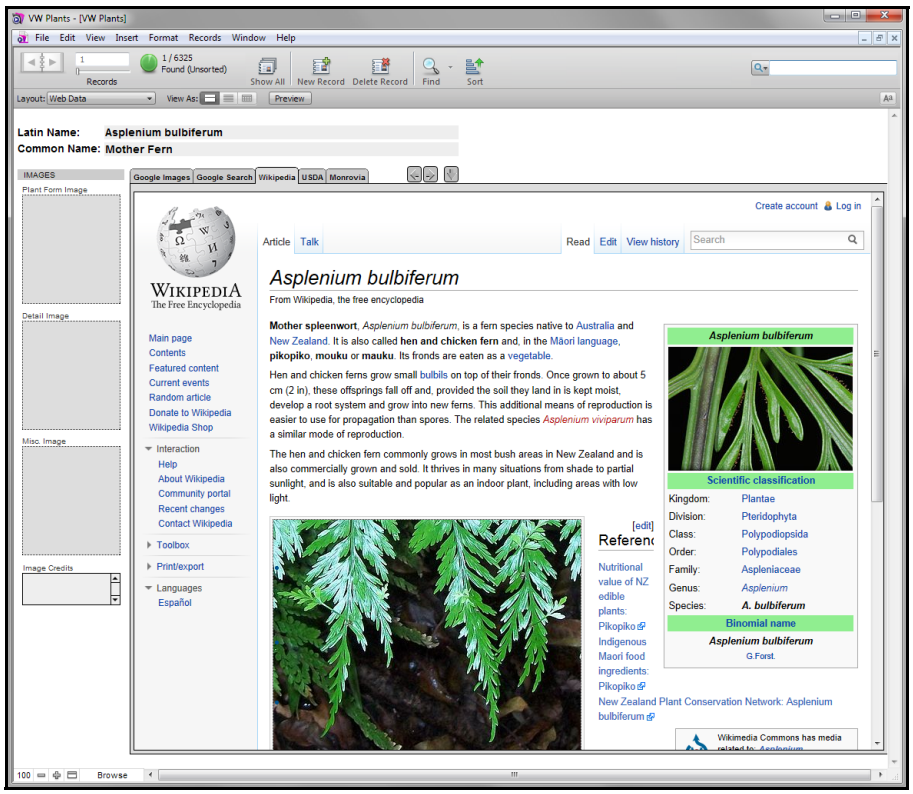
Accessing Plant Information from the Internet

If access to the Internet is available, plant images and information are easily obtained from within the plant database window. Images can be copied directly into the database (image credits can also be specified).

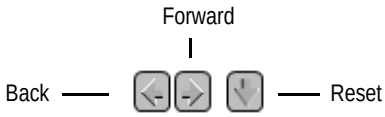
To access plant information from the Internet:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Locate the plant record that requires images or information.
3. Select the **View > Swap Data View/Web View** database command to toggle to web view, or select the Web Data layout from the Status toolbar.

   In web view, a search is automatically conducted for the current plant based on its Latin name, and information about it, as well as images, are displayed from different sources.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

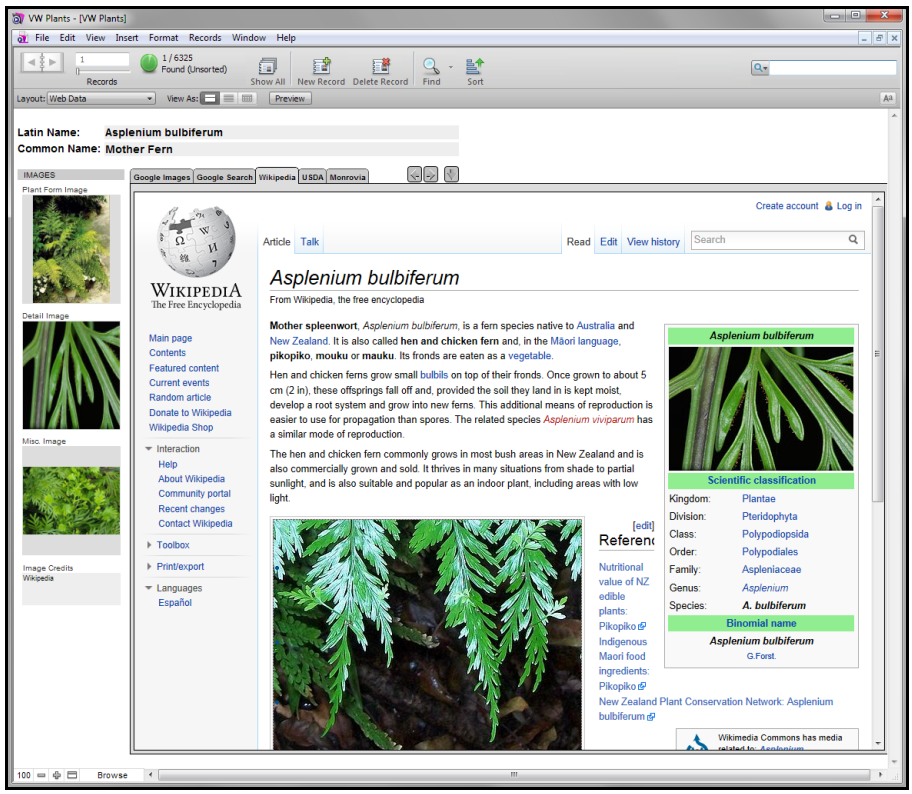


4. Click on the web view tabs to find plant information from the various sources. The available sources may depend on regional settings provided by a distributor.
5. Click the buttons to navigate through the web pages of each tab or to reset the view to the initial search (based on the Latin name).



6. To easily copy an image from the Internet to the plant database, select **Copy** (Windows) or **Copy Image** (Mac) from the image context menu. Then select **Paste** from the image container context menu. Image credits can be added in the Image Credits area under the images, to avoid copyright issues.

   Image files can also be saved and inserted into the database later, as described in “Editing Plant Records” on page 811.



7. Select the **View > Swap Data View/Web View** database command, or change the layout from the Status toolbar, to return to the database view.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Plant Database

Creating Plant Lists for Vectorworks Landmark

Plant lists are the connection between the FileMaker plant database and the Vectorworks Landmark product. Creating plant lists from the entire set, or from found sets, in the plant database allows that data to be attached to plants in the Vectorworks program. Create as many plant lists as necessary; the appropriate plant list is selected for the plant definition. The data attached to a plant becomes part of the plant definition, and can be added to planting plans and displayed by plant ID tags.

Plant lists are created from the plant database; plant lists can also be created automatically based on plant category.

To create a plant list from the plant database:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Search for the plants to include in the plant list (see “Searching for Plants” on page 808). The plant list is created from a found set of records, or from all the plant records. Larger plant lists take longer to load into the Vectorworks program.
3. Select the **File > Create Vectorworks Plant List** database command.

   The Create Plant List dialog box opens. Provide a name for the plant list.
4. Click **OK**.

   The plant list file is a tab-delimited file saved in the location of the plant database.

   Click here for a video tip about this topic (Internet access required).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Plant Lists from Categories

To create plant lists based on plant category:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Select the **File > Create Category Plant Lists** database command.

   A plant list is created for each category. If there are no plants found for a category, no list is created for that category. Plants without a category assigned, or with a custom category, are not included. In addition, an “All” plant list contains all plants regardless of category.
3. The plant lists are tab-delimited files saved in the location of the plant database.

Plant Lists for Existing Tree Species

By default, all plants in the tree **Category** of the database are included in the Species Data dialog box for access by the **Existing Tree** tool. If desired, search the plant database for the trees you require, and then add only these trees to the existing tree species list.

To create a custom existing tree species list:

1. Open the plant database as described in “Accessing the Plant Database” on page 804.
2. Search for the trees to include in the existing tree species list (see “Searching for Plants” on page 808).
3. Select the **File > Create ‘Existing Tree’ List** database command. This customizes the existing tree list to use only the plants found in the database search.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Plant Database
Specifying Existing Tree Species Information

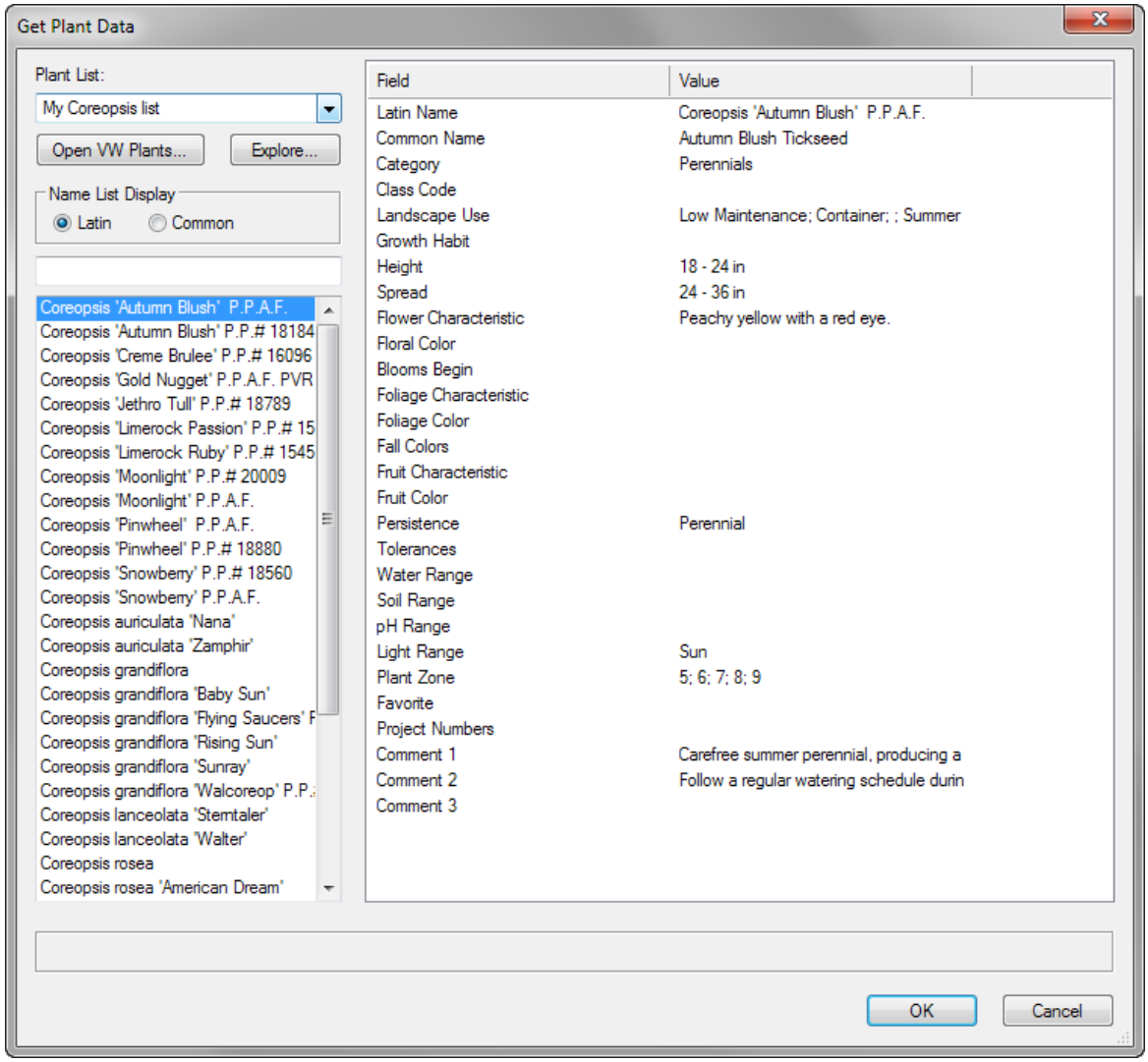
Using Plant List Data in Vectorworks Landmark

Plant list information is available to plant definitions from the Plant Data tab.

To use plant list data in a plant definition:

1. Create the plant definition as described in “Creating Plant Definitions” on page 787.
2. In the Edit Plant Definition dialog box, click **Get Plant Data**.

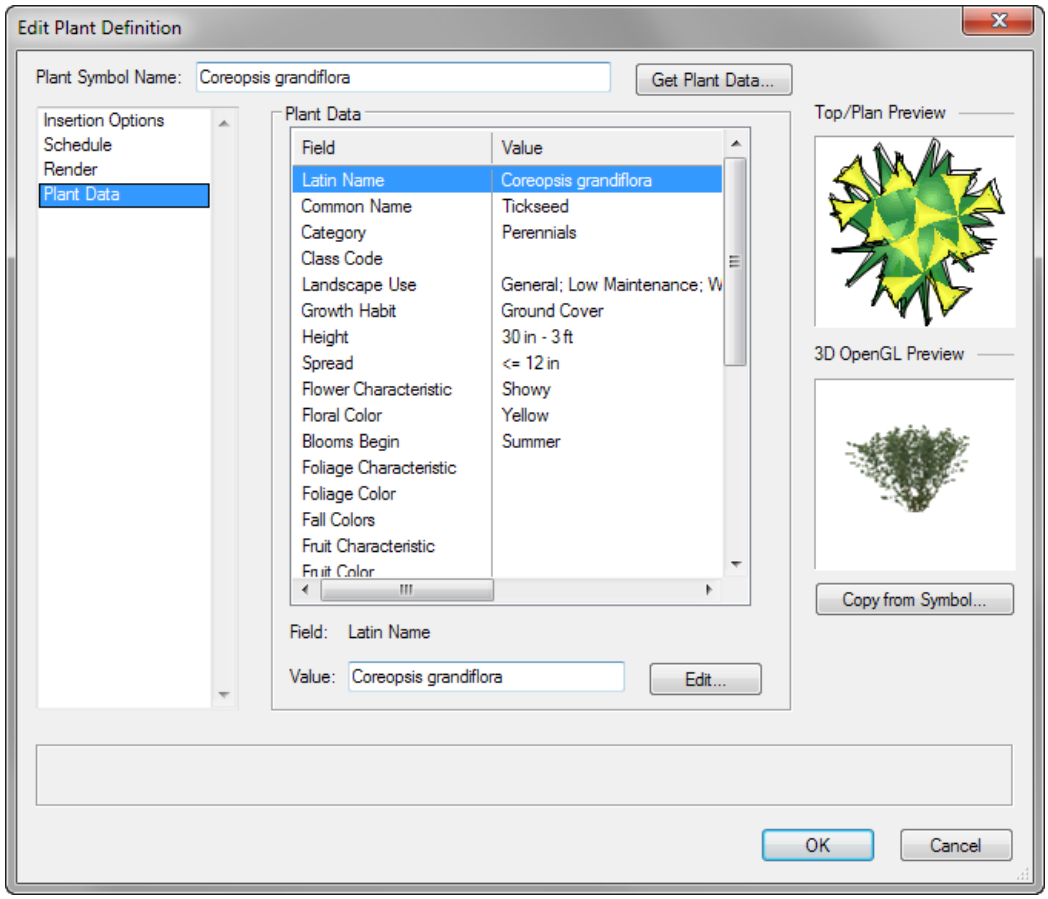
   The Get Plant Data dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click to show/hide the parameters.

Parameter	Description
Plant List	Select the plant list that contains the required plant information; all plant lists created from the plant database and saved in the database location are listed. A progress bar indicates the status of the plant list import when it is used for the first time, and allows the import to be canceled.
Open VW Plants	Opens the Vectorworks plant database, for locating plant information and creating plant lists
Explore	Opens Explorer (Windows) or Finder (Mac) to the folder containing generated plant lists
Name List Display	Toggles between plant list display by Latin name or common name
Name Find	Finds a plant in the plant name list by matching typed characters
Name List	Lists the plants included in the currently selected plant list. Select a plant to view its botanical information on the right, and to use that data in the plant definition.
Plant Data	Displays the botanical information (from the plant database) for the plant selected in the name list

3. Select the plant name and data to include in the plant definition, and click **OK**.
4. The Plant Data tab displays the data from the plant database.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Plant Lists for Vectorworks Landmark
Creating Plant Definitions

Plant Database Field Mapping

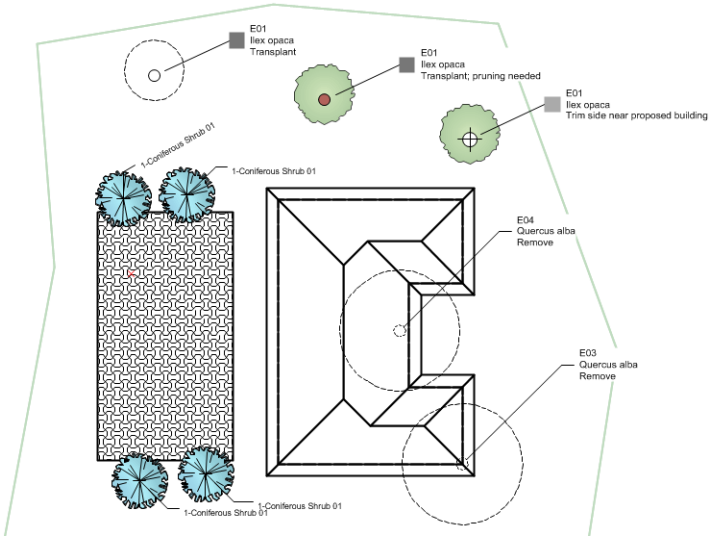
When plant data is imported into the plant database with the **File > Import Records > File** database command, field mapping is required (see “Importing Plant Database Information” on page 806). Certain fields must be used during mapping for the plant list to function properly when exported for use with the Vectorworks Landmark product. All significant Vectorworks fields have a VW prefix.

Target Field Name	Notes or Example Values	Target Field Name	Notes or Example Values
VW Autumn Color		VW Height	
VW Bloom Time	Also referred to as Season	VW Landscape Use	Border, Hedge, Shade Tree...
VW Category	Shrubs, Herbs, Trees...	VW Latin Name	Also known as the plant botanical name
VW Climate Zone	Arid, Semi Arid, Dry...	VW Light Range	Deep Shade, Shade, Sun, Full Sun
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Target Field Name	Notes or Example Values	Target Field Name	Notes or Example Values
VW Code		VW Other Tolerance	
VW Comments 1		VW Other Use	
VW Comments 2		VW Persistence	Deciduous, Semi-Evergreen, Evergreen...
VW Comments 3		VW pH Range	Acidic, Adaptable
VW Common Name		VW Price	
VW Favorites		VW Region	
VW Flower Characteristics	Double, Erect, Fragrant, Horizontal...	VW Region Notes	
VW Flower Color		VW Season Notes	
VW Foliage Characteristics	Aromatic, Broad-leaf, Evergreen, Fronds, Small leaves...	VW Size Notes	
VW Foliage Color		VW Soil Range	Bark, Sand, Sandy loam, Potting soil...
VW Fruit Characteristics	Acorns, Berry, Catkins, Cones	VW Spread	
VW Fruit Color		VW Tolerances	Cold Frost, Drought, Heat...
VW Growth Habit	Arching, Broad-domed, Columnar, Climber...	VW Used in Project	
VW Hardy Zone	1 – 11 (from USDA zone mapping)	VW User notes	

L Documenting Existing Trees

The Vectorworks Landmark product includes the **Existing Tree** tool, for documenting the existing trees on a site. Details about each tree can be provided, including information such as species, condition, life expectancy, significance, and action to be taken. The 2D and 3D appearance of the trees can be specified, indicating the root and tree protection zones, with detailed plant ID tags and graphics for retained and removed trees, and the ability to select a 3D trunk and canopy shape. A report including the relevant information can be automatically created.



Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Existing Tree Workflow

When placing existing trees into the drawing, the following workflow is recommended:

- Initialize the plant database if it has not yet been opened, by selecting **Landmark > Choose VW Plants**. See “Specifying Existing Tree Species Information” on page 824 and “The Plant Database” on page 804. This action populates the list of existing trees so that species information can be selected.
- Click the **Existing Tree** tool from the Site Planning tool set, and then select **Preferences** from the Tool bar to open the Object Properties dialog box for the existing tree. Set the defaults for the placement of existing trees, including automatic numbering, ID tag, 2D and 3D appearance (including the selection of 2D component symbols), and species information.
- While setting the default properties for the **Existing Tree** tool, specify classing wherever possible to be able to control the visibility of classed elements (easily accomplished from the Navigation palette) as well as specify appearance by class for many of the elements.
- Select **Tools > Organization** to open the Organization dialog box. On the Classes tab, navigate to the classes related to the existing tree, and specify their appearance. Classes should be set to **Use at Creation**. For canopy and trunk classes, set the texture (Renderworks required) or color to display in 3D views. Set the marker and line style of the tag class if a marker is to be displayed.
- When placing existing trees, keep in mind that the selected 2D component symbols will not display until an **Action** has been specified. This helps you determine at a glance which trees still require an action to be set.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- The parameters of groups of selected trees, or trees on a specific layer, or all trees can be set in a single operation by using the **Apply Properties** in the various existing tree dialog boxes.
- Indicate the location of important and less important areas of the site by specifying the significance of each tree or group of trees.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

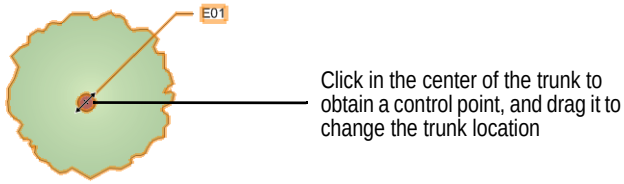
Documenting Existing Trees

Placing Existing Trees

To place an existing tree:

1. Click the **Existing Tree** tool from the Site Planning tool set.

   If this is the first time the object has been placed in this session, the Object Properties dialog box opens. Set the default existing tree properties before placing trees, or place the tree and then edit its parameters in the Object Info palette.
2. Click in the drawing to place the existing tree. Click again to set the rotation.

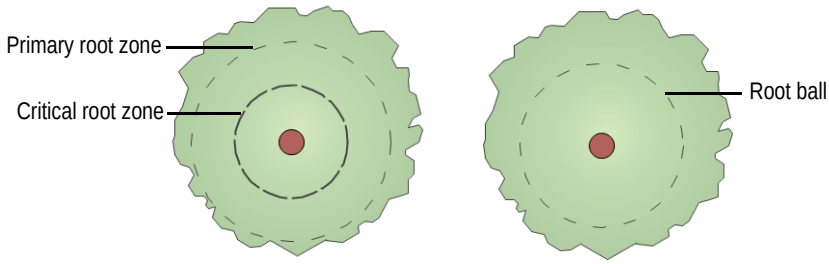


   The appearance of the existing tree may not match its 2D settings until certain parameters have been specified in the Object Info palette. For example, until the **Action** is specified, the tree does not display with the symbol set for Retain or Remove; the significance must be set for the significance color to display.

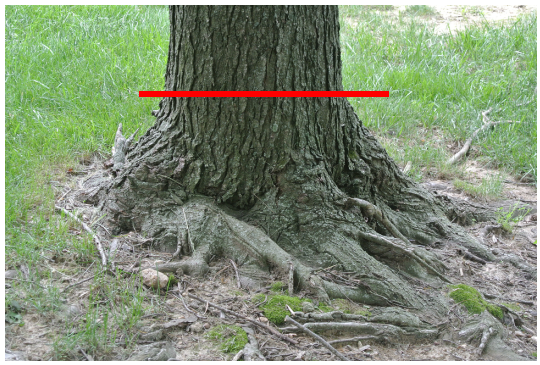
The existing tree properties can be edited from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Get Species Data	Opens the Species Data dialog box, to obtain botanical information from the plant database; see “Specifying Existing Tree Species Information” on page 824
Botanical Name/ Common Name	Displays the tree’s botanical and common name, when obtained by clicking **Get Species Data**; alternatively, enter the botanical and common name if not using the plant database information
Tag and Number Options	Opens the Tag and Number Options dialog box, to set the plant ID tag and automatic numbering options; see “Existing Tree Autonumbering and ID Tags” on page 831
Tree No	Displays the tree’s autonumber, when autonumbering is enabled; alternatively, if autonumbering is not selected, enter an ID number for the tree
Origin	Specifies whether the tree is an endemic, native, or exotic species
Height	Sets the height of the tree
Irregular Canopy Size	Opens the Irregular Canopy Size dialog box, to specify the plan shape of an irregular canopy rather than use the regular canopy diameter method; see “Specifying an Irregular Canopy” on page 828.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
North Angle (irregular canopy shape)	Rotates an irregular canopy to enable the North, East, South, and West dimensions of the canopy to be correctly aligned to the points of a compass
Canopy Max. Diam (regular canopy shape)	Sets the tree's maximum canopy diameter
Also Use Canopy Min. Diameter (regular canopy shape)	Enables the canopy minimum diameter option, which allows an additional smaller canopy size to be specified, for tree canopy shapes that are oval rather than circular
Canopy Min. Diam	Specifies the minimum dimension of the canopy; the oval canopy shape is created within the specified maximum and minimum values. The 3D solid shape is also affected.
Canopy Rotate (regular canopy shape)	Sets the rotation angle of the tree canopy; this is useful when the canopy is an elliptical or uneven shape
First Branch Hgt	Indicates the height of the lowest (first) branch; this determines the bottom position of the 3D canopy
DBH	Specifies the diameter of the tree trunk at breast height. Adult breast height is considered to be 51 – 55" (1.3 – 1.4 m) above the ground. The DBH value also determines the initial diameter of the critical and primary root zones. A tree with multiple stems displays with the number of stems after the DBH value, which cannot be edited. Because a this kind of tree does not have a single central trunk, the DBH is calculated differently, by calculating the square root of the sum of each stem's squared diameter value.
Multiple Stems	Opens the Multiple Stem Calculator dialog box, for trees with more than one central trunk or stem. Select **Use Multiple Stems**, and then specify the diameter of each stem or trunk to include in the DBH calculation. The DBH is automatically calculated, but can be adjusted if necessary.
Root Display	Sets whether to display the critical and/or primary root zones, the root ball zone only, or no root zones. Specify the appearance of the zones by clicking **2D Properties** (see "Setting 2D Existing Tree Appearance" on page 825). The root zones move with the tree trunk location. 
Critical Root Diam	Specifies the diameter of the critical root zone; initially, this value is set to 2.5 times the **DBH**, but the value can be changed. A changed value is reset if the **Root Display** is turned off; set the value to 0 (zero) to re-apply the **DBH** factor.
Primary Root Diam	Specifies the diameter of the primary root zone; initially, this value is set to 5 times the **DBH**, but the value can be changed. A changed value is reset if the **Root Display** is turned off; set the value to 0 (zero) to re-apply the **DBH** factor.

Parameter	Description
Ball Diameter	When the root ball is set to display, indicates the diameter of the root ball
Tree Protection	Select how to display the area around the tree that protects the tree; see “Specifying the Tree Protection Zone” on page 828
TPZ Cnr Radius (irregular zones)	When the tree protection zone editing mode is enabled for irregular protection zones, specifies the corner radius of the protection zone; set to a smaller value to simulate sharp corners
TPZ Radius (circular zones)	For a circular or 12 x DBH tree protection zone, displays the radius of the zone
TPZ Area	Displays the calculated area of the tree protection zone
Condition	Indicates the condition of the tree: excellent, good, average, poor, or custom
Custom Condition	When a custom condition is selected, enter the custom tree condition
SULE	Indicates the Safe Useful Life Expectancy (SULE) rating for the tree. A custom life expectancy value can be entered.
Custom SULE	When a custom SULE is selected, enter the custom life expectancy
Significance	Specifies the significance of the existing tree. This selection causes the significance color set in the 2D Properties dialog box to display. A custom significance can be set.
Custom Significance	When a custom significance is selected, enter the custom tree significance. To preserve the ability to set and display the significance color automatically, select one of the provided significance values, and then select Custom. Keep the initial default significance word, and then enter any custom information. For example, to use the Hazardous category, but add custom comments, select Hazardous, and then Custom. The word “Hazardous” displays for **Custom Significance**. After the word Hazardous, add your custom comment, such as “Hazardous - remove immediately.” By retaining the word Hazardous at the start of the custom comment, the significance color selected for Hazardous still displays.
Action/Comment	Indicates the action to be taken for the tree. The category selected (Retain, Transplant, or Remove) determines the symbol to display based on the selections made in the 2D Properties dialog box. A custom action can be set. When transplanting a tree, set the original tree **Action** to Transplant - Original location. Copy the tree, disabling its automatic numbering properties and manually assigning it to the same **Tree No.** as the original tree. Move the copy to the new tree location and set the **Action** for the copy to Transplant - Proposed location.
Custom Action	When a custom action is selected, enter the custom action to take for the tree. To preserve the ability to automatically set the displayed symbol, select one of the provided actions, and then select Custom. Keep the initial default action category, and then enter any custom information. For example, to use the Remove category, but add custom comments, select Remove, and then Custom. The word “Remove” displays for **Custom Action**. After the word Remove, add your custom comment, such as “Remove - storm damaged.” By retaining the word Remove at the start of the custom comment, the symbol selected for Remove still displays.
Additional arborist fields	These specialized fields only display when enabled in the Additional Fields dialog box; each tree can be rated according to whether it meets arborist criteria

Parameter	Description
Form	This refers to the shape of the tree, from uniform (excellent) to lopsided (poor)
Structure	Trees with excellent structural integrity, growing in native habitat, rate higher than less stable trees or trees not growing in typical habitat for the tree type
Vigor	This describes the health of the tree and whether it is growing well, with no pests and with good structure and form
Diam. at Buttress	The diameter at buttress is the diameter of the trunk measured immediately above the base of the tree where the trunk mostly becomes parallel, as shown by the red line in this example: 
Show Structural Root Zone	The Structural Root Zone (SRZ) refers to the circular area around the base of a tree that is required for the structural stability of the tree. The appearance of the SRZ indicator is set by the Critical Root Zone parameters (**Critical Line Color**/**Style** in the 2D Properties).
Calc'd SRZ Radius	Displays the structural root zone radius value in meters, which is calculated based on the diameter at buttress value (D) according to the following formula: $\text{SRZ radius (m)} = (D \times 50)^{0.42} \times 0.64$ If the calculated value is less than the **Minimum SRZ radius** specified in the Additional Fields dialog box, the minimum value is displayed instead.
Reduced Level/Age/ Year Planted/Date Assessed/Location/ Notes	These parameters provide additional information about the tree and can be included in reports. Notes can be selected for display in the tree tag. The field names can be edited, and additional fields can be added, by clicking **Additional Fields**.
2D Properties	Opens the 2D Properties dialog box, to set display options for the 2D symbol components, significance colors, and the root zone and tree protection zone attributes; see "Setting 2D Existing Tree Appearance" on page 825
3D Properties	Opens the 3D Properties dialog box, to set the 3D canopy and trunk display options; see "Setting 3D Existing Tree Appearance" on page 829
Additional Fields	Opens the Additional Fields dialog box, for customizing Object Info palette fields, adding custom fields, and including arborist parameters to worksheet reports

Parameter	Description
Custom Field Labels and Visibility	The first six default fields from the Object Info palette can be edited to use different field names, and four additional fields can be named. Regardless of the names used for the fields, the worksheet formulas for displaying them remain fixed; mouse over each field to see its associated formula in the help text. • Field 1: `=("Existing Tree"."RL")` • Field 2: `=("Existing Tree"."Age")` • Field 3: `=("Existing Tree"."Year Planted")` • Field 4: `=("Existing Tree"."DateAssessed")` • Field 5: `=("Existing Tree"."Location")` • Field 6: `=("Existing Tree"."Comments")` • Field 7 – Field 10: `=("Existing Tree"."Fieldx")` where x is the field number
Display	Select how many fields to display in the Object Info palette
Additional Arborist Fields	Displays specialized arborist parameters in the Object Info palette
Display Form, Structure, and Vigor	Displays these fields in the Object Info palette
Display Buttress, Structural Root Zone, etc.	Displays these fields in the Object Info palette
Minimum SRZ radius	Specifies the minimum required Structural Root Zone (SRZ) radius; if the calculated SRZ radius is less than this value, this value is used instead
Apply Properties	The custom fields can be applied to a single tree, to all trees in the drawing, and as the tool default settings. • This object only: Applies the properties to the currently selected existing tree • All objects in file: Applies the properties to all existing trees in the drawing • This object + file defaults: Applies the properties to the currently selected existing tree and sets the properties as the default properties for the **Existing Tree** tool. Existing trees placed by the tool after clicking **OK** will have these properties applied. • All objects in file + file defaults: Applies the properties to all existing trees in the drawing and sets the properties as the default properties for the **Existing Tree** tool. Existing trees placed by the tool after clicking **OK** will have these properties applied.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Documenting Existing Trees

Specifying Existing Tree Species Information

The species information for existing trees is obtained from the Vectorworks Landmark plant database; see “The Plant Database” on page 804. If the plant database has not yet been opened, its default location needs to be set.

To specify botanical data from the Vectorworks plant database:

1. If it has not yet been opened, initialize the plant database by selecting the **Choose VW Plants** command from the appropriate menu:
   - Designer workspace: **AEC > Plants > Choose VW Plants**
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Landmark workspace: **Landmark > Choose VW Plants**.

The Choose Plant Database Location dialog box opens. Select the plant database location as described in “Accessing the Plant Database” on page 804.

By default, all plants in the tree **Category** of the database are included in the Species Data dialog box for access by the Existing Tree tool. If desired, search the plant database for the trees you require, and then select the **File > Create ‘Existing Tree’ List** database command. This customizes the existing tree list to use only the plants found in the database search.

2. From the Object Info palette of a selected existing tree, click **Get Species Data**.

   The Species Data dialog box opens, listing tree species information. Species can be easily added to the list.

   If the list is blank, the plant database location was not initialized.

Click to show/hide the parameters.

Parameter	Description
Species list	Lists the botanical and common name of the trees contained in the existing tree list; click either heading to sort. Black text indicates species that appear in the Existing Trees.txt file located in the Plant Database folder within the user folder. Blue text indicates species that you have added to the list.
Botanical Name/ Common Name	Displays the botanical and common name of the selected tree; to add a tree to the list, enter its botanical information and click **Add Tree to Existing Tree List**
Remove Tree from Existing Tree List	Removes the selected custom (blue text) tree species from the list. Black text indicates default species, which cannot be removed from the existing tree list.
Add Tree to Existing Tree List	Adds the current **Botanical Name** and **Common Name** to the existing tree list. Edit a default entry if needed, or add a new species to the list. Custom (added or edited) species display with blue text.

3. Select the species from the list, and click **OK**.

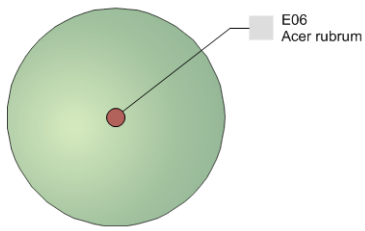
   The Object Info palette displays the species information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Documenting Existing Trees

Setting 2D Existing Tree Appearance

The 2D appearance controls the symbol for the tree and trunk, and also sets the attributes of the significance status, primary and critical root zones, and tree protection zones. The settings apply to the selected tree or a variety of other selection options, and can be set as the default 2D attributes for the drawing. This is a powerful way to control the 2D appearance of all existing trees in the layer or drawing in one operation.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using classes for the significance colors, root zone, and tree protection zone is recommended. The visibility of these elements is easy to control with classes. See “Classes” on page 176.

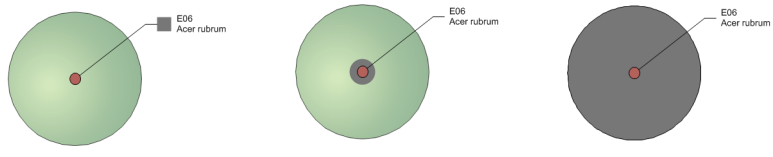
Existing tree symbols are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219). The resources in the ET Symbols.vwx file are available in the 2D Properties dialog box.

To set the existing tree 2D appearance:

1. From the Object Info palette of a selected existing tree, select **2D Properties**. Alternatively, double-click on an existing tree.

   The 2D Properties dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Use 2D Component Symbols	Uses 2D symbol components to display the tree. The tree symbol displayed in the drawing depends on the **Action** specified in the Object Info palette.
Tree Retain	Select the symbol to display for a tree that is retained (the associated Object Info palette **Action** is Retain)
Tree Remove	Select the symbol to display for a tree to be removed (the associated Object Info palette **Action** is Remove)
Tree Trunk	Select the tree trunk symbol for either retained or removed trees
Transplanted Trees	Select the symbol to display for the existing position of a tree to be transplanted (the associated Object Info palette **Action** is Transplant - Original location); select the symbol to display for the new position of the transplanted tree (the associated Object Info palette **Action** is Transplant - Proposed Location). Specify the tree trunk symbol for transplanted trees in both the original and new locations.
Use Significance Colors	Applies significance colors to the tree to indicate its relative importance on the site. The significance color used depends on the **Significance** selected in the Object Info palette.
Significance rating colors	For each significance rating, select the color to display. For more information on colors, see “Applying Colors” on page 1132.
Display	Specifies where to place the significance color.  Indicator beside tree ID Circle around trunk Tree fill color Use the significance color as a tree fill for complex plans with multiple trees, to easily distinguish significance categories for densely forested areas.
Class	If desired, select a class to control the display of the significance indicators. The classes present in the drawing are listed, or select the default class, Exist Tree-Significance. Class attributes control all indicator attributes with the exception of the fill color (any fill colors selected here override the class settings).
Root Zone Attributes	Applies line colors and styles to the primary and critical root zone indicators. Root zones must be selected for display in the Object Info palette.

Parameter	Description
Primary Line Color/ Style	Select the color and line style for the primary root zone line
Critical Line Color/ Style	Select the color and line style for the critical root zone line
Class	If desired, select a class to control the display of the root zone indicators. The classes present in the drawing are listed, or select the default class, Exist Tree-Root Zones. The line attributes set here override the class settings.
Tree Protection Attributes	Applies a line color and style to the tree protection zone indicator. The tree protection zone must be selected for display in the Object Info palette.
TPZ Line Color/ Style	Select the color and line style for the tree protection zone line
Class	If desired, select a class to control the display of the tree protection zone indicator. The classes present in the drawing are listed, or select the default class, Exist Tree-TPZ. Any line attributes set here override the class settings.
Apply Properties	The various 2D properties can be applied to a single tree, all trees in the layer or drawing, and as the tool default settings
Apply list	Select the method to apply the properties. • To this object only: Applies the properties to the currently selected existing tree • To all objects on all layers: Applies the parameter categories selected in **Properties to Apply** to all existing trees in the drawing • To all objects on active layer only: Applies the parameter categories selected in **Properties to Apply** to all existing trees in the current layer • To all selected objects on all layers: Applies the parameter categories selected in **Properties to Apply** to selected existing trees in the drawing • To all selected objects on active layer only: Applies the parameter categories selected in **Properties to Apply** to selected existing trees in the current layer
Also apply as the document defaults	Sets the selected 2D properties as the default properties for the **Existing Tree** tool. Existing trees placed by the tool after clicking **OK** will have these properties applied.
Properties to Apply	When applying properties to more than just the currently selected tree, select which properties to globally apply

2. Click **OK**.

   Certain 2D properties depend on selections made in the Object Info palette for each existing tree, and are not applied immediately. For example, the symbol specified in **Tree Remove** does not display unless Remove is selected as the **Action** for the tree.

   The symbols for retained and removed trees are scaled by the canopy height and diameter specified in the Object Info palette. The original and proposed symbols for a transplanted tree are scaled by the **DBH** parameter.

Documenting Existing Trees
Applying Colors

Specifying the Tree Protection Zone

The tree protection zone is the area around the tree that should be fenced off, or preserved, to protect the tree. The tree protection zone moves with the tree trunk location. The zone can be irregular or circular; irregular zones can be reshaped. The **TPZ Area** displays the calculated area for the entire tree protection zone.

To specify the tree protection zone:

From the Object Info palette of a selected existing tree, select the type of tree protection zone.

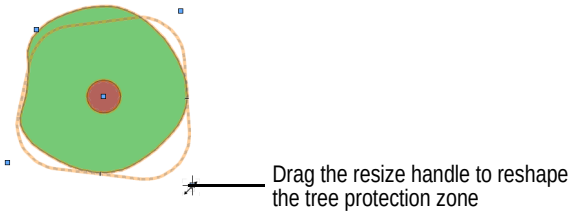
Click to show/hide the parameters.

Zone	Description
None	Does not display a tree protection zone
Display Irregular Zone	Displays the tree protection zone as an irregular shape which can be reshaped as needed to represent the zone
Edit Irregular Zone	Allows an irregular zone to be reshaped as described below
Display Circular Zone	Displays the tree protection zone as a circle around the tree trunk; set the circle radius with **TPZ Radius**
Display using 12x DBH	Displays a circular tree protection zone that is calculated based on 12 times the diameter at breast height (**DBH**) parameter

An irregular tree protection zone may need to extend beyond the initial boundary shape. To reshape the tree protection zone:

1. Select Edit Irregular Zone from the **Tree Protection** parameter.

   The tree protection zone displays with editing handles.



2. Drag the handles to reshape the zone. Adjust the radius at the corners of the zone with the **TPZ Cnr Radius** value, which only displays during zone editing.
3. When reshaping is finished, select Display Irregular Zone from the **Tree Protection** parameter.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Documenting Existing Trees

Specifying an Irregular Canopy

Since trees are not always perfectly symmetrical, the **Existing Tree** tool allows an irregular canopy shape to be defined by setting the measurements, in four different directions, from the center of the trunk to the edge of the canopy.

If the trunk had previously been repositioned by moving its control point, it is reset to the default position when an irregular canopy is selected. The trunk cannot be offset for irregular canopy shapes.

To specify an irregular canopy shape:

1. From the Object Info palette of a selected existing tree, select **Irregular Canopy Size**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Irregular Canopy Size dialog box opens. Specify the canopy size in four directions.

Click to show/hide the parameters.

Parameter	Description
Use Canopy Offsets from Center of Trunk	Select to specify an irregular canopy shape
North/East/South/West	For each direction, specify the distance from the center of the trunk to the edge of the canopy

2. Click **OK**. The 2D and 3D appearance of the tree displays with an irregular canopy shape, roughly based on the selected symbol definition.

   The irregular shape can be oriented differently by entering a **North Angle** offset in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Documenting Existing Trees

Setting 3D Existing Tree Appearance

The 3D appearance controls the symbol for the tree and trunk in 3D views. The settings apply to the selected tree or a variety of other selection options, and can be set as the default 3D attributes for the drawing. This is a powerful way to control the 3D appearance of all existing trees in the layer or drawing in one operation. Showing trees in a 3D view is also useful for creating shadow diagrams (Renderworks required).

Color shader texture combined with noise transparency shader

Using classes for the tree canopy shape and trunk is recommended. The attributes (color or texture) of these elements are easy to control with classes (Renderworks is required to use textures). The class color or texture must be set to **Use at Creation**. See “Classes” on page 176.

Existing tree symbols are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219). The resources in the ET Symbols.vwx file are available in the 3D Properties dialog box.

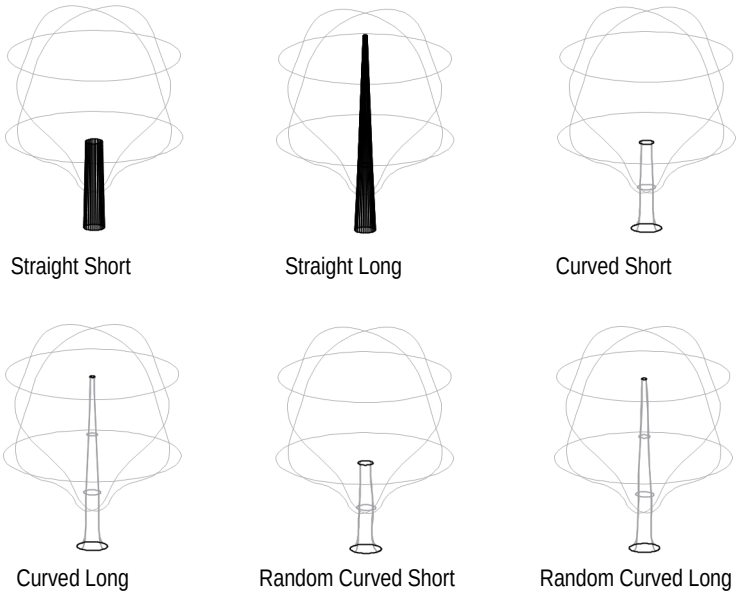
Image props, such as those provided by xFrog, can be used as 3D tree geometry. However, canopy height and canopy diameter changes in the Object Info palette do not affect the geometry of image props.

To set the existing tree 3D appearance:

1. From the Object Info palette of a selected existing tree, select **3D Properties**.

   The 3D Properties dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Create 3D Geometry	Select whether to create 3D geometry for the existing tree
Canopy Shape	Select the symbol to display for the 3D canopy
Trunk Shape	Select the symbol to display for the 3D trunk shape. Short trunks extend to the **First Branch Hgt** value indicated in the Object Info palette, while long trunks extend into the canopy. If using a canopy texture that is partially transparent, use a long trunk shape for a more realistic appearance. A curved shape gives the trunk a flanged appearance near the ground; random selections are slightly more free-form in shape. 
Optional Class Names	
3D Canopy	If desired, select a class to control the appearance and visibility of the canopy. The classes present in the drawing are listed, or select the default class, Exist Tree-3D Canopy.
3D Trunk	If desired, select a class to control the appearance and visibility of the trunk. The classes present in the drawing are listed, or select the default class, Exist Tree-3D Trunk.
Apply textures to classes (Renderworks required)	When the canopy and/or trunk appearance is defined by classes, determines whether to apply the texture set for the class. In drawings with many external trees, performance may be improved by turning off the class texture. As the project becomes complete, apply the classes for optimal presentation.
Apply Properties	The various 3D properties can be applied to a single tree, all trees in the layer or drawing, and as the tool default settings

Parameter	Description
Apply list	Select the method to apply the properties. • To this object only: Applies the properties to the currently selected existing tree • To all objects on all layers: Applies the properties to all existing trees in the drawing • To all objects on active layer only: Applies the properties to all existing trees in the current layer • To all selected objects on all layers: Applies the properties to selected existing trees in the drawing • To all selected objects on active layer only: Applies the properties to selected existing trees in the current layer
Also apply as the document defaults	Sets the 3D properties as the default properties for the **Existing Tree** tool. Existing trees placed by the tool after clicking **OK** will have these properties applied.

2. Click **OK**.

   The 3D canopy shape is affected by changes in the Object Info palette for height, diameter, irregular canopy shape, **First Branch Hgt**, and **DBH** values.

Documenting Existing Trees

Existing Tree Autonumbering and ID Tags

Existing tree tags can contain an automatic numbering ID as well as tree information. Set up the new tree autonumbering properties first, to automatically number the trees according to the settings as they are placed on the drawing. Autonumbering can also be changed after placement on a tree-by-tree basis.

To set autonumbering and ID tag information for existing trees:

1. From the Object Info palette of a selected existing tree, select **Tag and Number Options**.

   The Tag and Number Options dialog box opens.

Click to show/hide the parameters.

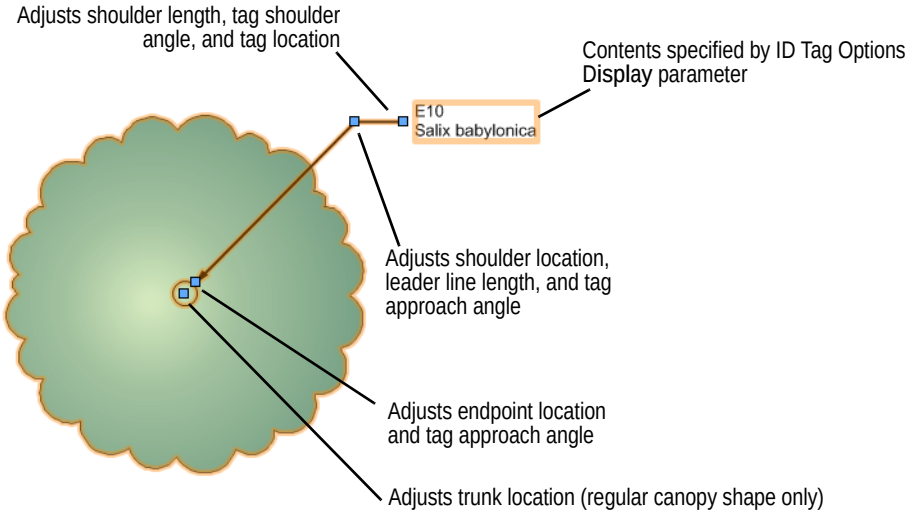
Parameter	Description
Auto Number New Trees	Sets autonumbering defaults for trees to be placed on the drawing. Deselect the option if autonumbering is not desired.
Leading Zeros	Specify the number of leading zeros to use for the autonumber. Using leading zeros allows correct sorting in worksheets.
New Tree Prefix/ Suffix	Specifies a prefix and or/suffix, if any, before and after the autonumber
Auto number duplicated trees	When selected, existing trees that are copied or duplicated (with the **Duplicate** command, or by clicking and dragging with the Option (Mac) or Ctrl (Windows) key pressed) receive their own autonumber
Separate numbering on each layer	Restarts the autonumbering series for each layer of the drawing

Parameter	Description
Auto Number This Tree	Sets autonumbering properties for the currently selected tree; deselect to manually enter a number in **Full ID Number** instead of using an autonumber. When displaying transplanted trees, for example, the same number is needed for the tree representation in the original and new locations. For one of the representations, deselect **Auto Number This Tree** and enter the identical number in the **Full ID Number**. If **Auto number duplicated trees** is not selected, **Auto Number This Tree** is automatically disabled when an autonumbered tree is duplicated.
Leading Zeros	Specifies the number of leading zeros to use for the autonumbering of the selected tree
This Tree Prefix/ Suffix	Specifies a prefix and or/suffix, if any, before and after the autonumber for the selected tree
Full ID Number	Displays the selected tree's autonumber; deselect **Auto Number This Tree** to specify a custom ID Number
ID Tag Options	Specifies how to display existing tree ID tags
Snap tag to edge of trunk	Snaps the end of the leader line to the edge of the trunk; deselect to adjust the endpoint location manually
Display marker at end of tag leader	Adds a marker to the leader line endpoint. The marker style is controlled by the ID tag class.
Enable tag shoulder line	Adds a shoulder to the leader line
Shoulder Angle	Sets the angle of the ID text and shoulder
Display	Specifies the information to display in the tag; each item displays on its own line. The **Notes** information is set in the Object Info palette.
Class	If desired, select a class to control the appearance (color of text and attributes of the leader line, as well as marker style) and display of the ID tag. The classes present in the drawing are listed, or select the default class, Exist Tree-Tags.
Apply To	Select the method to apply the numbering and ID properties. • To this object only: Applies the properties to the currently selected existing tree • To all objects on all layers: Applies the properties to all existing trees in the drawing • To all objects on active layer only: Applies the properties to all existing trees in the current layer • To all selected objects on all layers: Applies the properties to selected existing trees in the drawing • To all selected objects on active layer only: Applies the properties to selected existing trees in the current layer
Also apply as the document defaults	Sets the tag and numbering properties as the default properties for the **Existing Tree** tool. Existing trees placed by the tool after clicking **OK** will have these properties applied.
Also apply position of ID tag	Also applies the position of the ID tag and leader line from the currently selected tree to the existing trees set in **Apply To**
Full ID Number	If **Auto Number This Tree** is not selected, manually enter a number for the selected tree

2. Click **OK**.

Existing tree tag appearance and placement is flexible, and can be adjusted in several ways, including through the Object Info palette, tag class settings, and the control point locations on the drawing.

- When existing tree tags are required, define their appearance in the Tag and Number Options dialog box. Apply the ID tag options to all existing trees and to the document defaults.
- Individual tags can then be changed for selected existing trees by clicking **Tag and Number Options** from the Object Info palette. For example, selected trees in one area of a planting plan look more uniform when they all use the same tag approach and tag shoulder angle. Set the **Shoulder Angle** and then select **Also apply position of ID tag**.
- The existing tree tag class controls the appearance of the leader/shoulder lines, as well as the marker style.
- To move the plant tags of several selected plants at once, click the Unrestricted Interactive Scaling mode of the **Selection** tool.
- If an individual tag needs to be repositioned, plant tags also have several control points for adjusting the tag text and leader line and shoulder position and angle.
- Align and distribute the leader lines for improved readability with the **Align/Distribute Leader Lines** command (see “Aligning and Distributing Leader Lines” on page 1035)



Documenting Existing Trees

Customizing Existing Trees

Existing trees are hybrid symbols, containing a 2D symbol component, and optionally, a 3D symbol component. The default existing tree symbols are located in the default content included with the Vectorworks Landmark product in Vectorworks\Libraries (see “Resource Libraries” on page 219). As 2D and 3D settings are made, the associated tree symbols are automatically imported into the current file and appear in the Resource Browser.

As with any resource, the symbols can be edited through the Resource Browser, and these edits apply to the current file. Alternatively, custom existing tree symbols can be added to the default ET Symbols.vwx file, either directly or by exporting edited tree symbols to the file (duplicate and rename the symbols from the Resource Browser first, so that the original symbols are not overwritten). For more information on symbol operations in the Resource Browser, see “Working with Resources” on page 225. See “Exporting Custom Resources” on page 234 for information on exporting resources.

These naming conventions ensure that any custom symbols are available for selection in the 2D and 3D Properties dialog boxes:

- Symbol names for retained trees, or the new location of transplanted trees, must begin with **1.** (the number 1, followed by a period)
- Symbol names for removed trees, or the original location of transplanted trees, must begin with **2.**
- Symbol names for the 2D tree trunk must begin with **3.**
- Symbol names for the 3D canopy must begin with **4.**

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Symbol Definitions
Using the Resource Browser
Documenting Existing Trees

Plant Graphics

In addition to plant placement with the **Plant** tool, there are additional ways of representing plant masses on a planting plan. Large, defined planting areas containing specific plants can be created with the **Landscape Area** tool. Undefined plant masses and groups can be added with the **Plant Line** and **Vegetation Line** commands. To further change the appearance of the planting plan, all plant styles can be displayed or hidden.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Landscape Areas
Creating a Plant Line
Creating a Vegetation Line
Show/Hide Plant Styles

Creating Landscape Areas

Landscape areas consist of large, defined regions of plant combinations, useful for conceptual planning and specifying large planting or reforestation areas, when individual plant symbols do not need to be drawn. Plant information from a landscape area is based on plant symbol data, and is included in plant list worksheets. If a site model is present, the landscape area is calculated based on the site model and displays in the Landscape Area Settings dialog box.

A landscape area can also be created without plant information specified, for use as a general ground cover or mulch area with a label showing the total area required.

Landscape area objects can also be created by drawing a polyline and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

 To create a landscape area:

1. Click the **Landscape Area** tool from the Site Planning tool set.
2. Click **Preferences** from the Tool bar to specify or change the default landscape area parameters.

   The Landscape Area Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Tag Information	Specifies the landscape area tag format
Name	Provides a name for the landscape area, which can be displayed in the tag

Parameter	Description
Tag Display	Select On to display a landscape area tag to the right or left of the leader line; set the left or right position with the **Tag Text Angle**
Tag Class	Specifies the class for the landscape area tag, or allows a new class to be created for the tag
Tag Line Angle	Specifies the angle of the leader line, from 0 to 360°
Tag Text Angle	When a landscape area tag is set to display, sets the tag text display angle, from 0 to 360°. To display the area tag to the left, specify an angle greater than 90 or less than 270 degrees. To display the area tag to the right, specify an angle less than 90 or greater than 270 degrees.
Tag Header	Select the information to display in the first line of the landscape tag, or select **None** to hide the top portion of the ID tag; select **Set Custom Tag** to define a custom landscape area tag (see "Creating a Custom Landscape Area Tag" on page 838)
Tag Body	Select the information to display in the main portion of the landscape tag, or select **None** to hide this portion of the ID tag; select **Set Custom Tag** to define a custom landscape area tag (see "Creating a Custom Landscape Area Tag" on page 838)
Area Units	Select the units for calculating the landscape area; this is set to the file units by default (see "Units" on page 68), but can be changed
Display Tag Line Marker	Displays a marker at the end of the leader line; specify the marker to use by editing the tag class properties (see "Setting Class Properties" on page 179)
Graphics	
3D Display	Select a 3D display option for the landscape area; in 3D views, plants conform to the surface of the site model if one exists
2D Only	Does not create a 3D representation of the landscape area; the landscape area displays in top/plan view only
3D Plants	When the landscape area contains plant symbols with an image prop 3D component, switching to a 3D view displays the image props in a rectangular array, for a realistic 3D view. Plant density is determined by the distribution or percentage specified for the landscape area. 3D loci are placed when the plant definitions do not have a 3D component.
3D Poly	Creates a 3D polygon at a default elevation of 0, and can display a fill color when rendered. Alternatively, assign the landscape area a texture from the Render tab of the Object Info palette (Renderworks required)
Texture Bed	Creates a texture bed for the 3D representation of the landscape area. If there is more than one site model in the file, the texture bed conforms to the site model that is in the same layer as the landscape area, if there is one. Apply a texture to the landscape area (Renderworks required). Select the site model and click **Update** from the Object Info palette; the landscape area displays as a texture bed on the site model
Border Style	Specifies the type of border for the edge of the landscape area
Standard	Sets the landscape area border to a single line
Plant Cloud	Draws the landscape area border with a convex or concave cloud shape; specify the cloud billow type and the billow parameters. In addition to selecting the billow size and height, the billow variability sets the variation between the smallest and largest billows.
Plant Line	Creates a freehand plant line border for the landscape area; for a rougher plant line border, specify a short segment length with high segment thickness

Parameter	Description
Plant Information	Specifies how to calculate the plant quantities of the combined plants in the landscape area, for tag display as well as planting plan worksheets; select **Distribution Rate** or **Percentage**
Distribution Rate (automatic coverage area %)	Calculates plant quantity based on plant distribution rate. As each plant is added to the list of plants in the landscape area, its rate per area or from the center is specified; the percentage of each plant in the landscape area is calculated automatically.
Distribution Rate (custom coverage area %)	Calculates plant quantity based on plant distribution rate. As each plant is added to the list of plants in the landscape area, its rate per area or from the center is specified, as well as the percentage of each plant in the landscape area. This option allows plants to be merged within the landscape area.
Percentage	Calculates plant quantity based on plant percentage of the total. As each plant is added to the list of plants in the landscape area, its percentage is specified. The plant distribution rate must be the same for all plants in the landscape area; specify the rate and units to use.
Landscape Area	Displays the landscape area value based on the surface area of the site model found if one exists, or based on the landscape area polyline if no site model is present. The units for the display are specified by **Area Units**.
Plant list	Displays the plants included in the landscape area, including each plant's distribution rate and percentage of the plant total. The plant order can be changed by click-dragging an item in the # column to a new position; this affects the plant list order in the plant tag body.
Add	Click to open the Edit Plant Information dialog box, to add a plant to the list of plants in the landscape area, and specify its distribution rate or percentage
Edit	Click to open the Edit Plant Information dialog box, to edit the selected plant; a different plant symbol can be selected, or the plant distribution rate or percentage changed
Delete	Click to delete the selected plant from the list of plants in the landscape area

3. Specify the plants included in the landscape area, as well as their frequency. Click **Add** to specify or **Edit** to change an included plant and its distribution rate or percentage value. The total percentage of all plants in the landscape area should be 100 when finished.

   The Edit Plant Information dialog box opens.

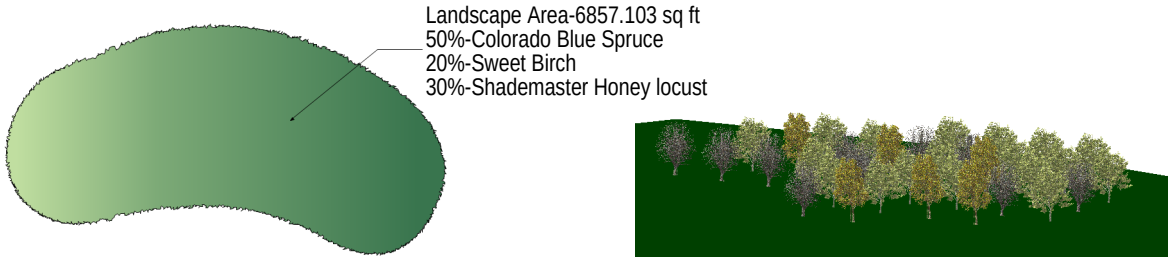
Click to show/hide the parameters.

Parameter	Description
Plant symbol list	To include plant information in the landscape area, select a symbol to insert from the graphical list. The list displays the symbols in the current document or selected plant content file; select a different plant content file from the list browser to display additional options.
Latin Name/Common Name/Plant/Tag ID	Displays the selected plant's definition information
Rate	If **Distribution Rate** was selected in the Landscape Area Preferences dialog box, indicate the plant distribution value and units. This distribution value is used to calculate the plant quantity for each plant type in the landscape area.

Parameter	Description
Percentage	If **Percentage** was selected in the Landscape Area Preferences dialog box, indicate the percentage of this plant in the landscape area. Using a fixed distribution rate for all plants, the percentage is used to calculate the plant quantity for each plant type in the landscape area. The total value of all plants must add up to 100%.
Plant Content File list	From the list browser, select the plant content file that contains the plant definition to insert; plant definitions in the selected plant content file are used to populate the plant symbol list. The Vectorworks Landmark software is installed with a basic selection of default plant definitions to get you started. Select **Help > Download Content** to download many more plant definitions in the free content libraries provided and/or add your own custom plant content (see "Resource Libraries" on page 219). Once added to the library, the appropriate plant content files automatically appear in the list browser. The list browser displays content located in [Vectorworks or User]/Libraries/Objects-Landscape & Site/Plant Tool-Symbol+Definition Library; check the filepath if the expected content does not display.

4. Click **OK**, and then click **OK** again to close the Landscape Area Preferences dialog box.
5. Click on the appropriate mode in the Tool bar to select the boundary creation method of the landscape area.

   For more information on the **Polyline** tool modes, see "Creating Polylines" on page 298.
6. Click to set the landscape area's start point.
7. Click to set the end of the segment and the beginning of the next. Continue drawing segments in this manner until the landscape area object is complete.



2D landscape area

3D landscape area with 3D plants

The appearance of the landscape area is controlled by several methods.

- Change the landscape area parameters from the Object Info palette, including specifying a general unit price (for indicating the price per square unit in worksheets), price code (such as a SKU number) and vertex parameters; click **Landscape Area Settings** from the Object Info palette to change landscape area information and other preference parameters
- Change the landscape area 2D attributes from the Attributes palette
- Change the landscape area tag appearance by editing its parameters in the Object Info palette, editing the tag class properties, and/or moving tag control points (similar to a plant tag; see "Plant Tag Appearance" on page 800)
- To align landscape area tags for improved readability, use the **Align/Distribute Leader Lines** command (see "Aligning and Distributing Leader Lines" on page 1035).
- Reshape the landscape area by double-clicking on it. The **Reshape** tool is automatically activated, to reshape the object directly in the drawing.

- Right-click (Windows) or Ctrl-click (Mac) on the landscape area object and select **Edit** from the context menu. The Edit Landscape Area dialog box opens. Either edit the settings of the selected landscape area, or edit the shape of the object path with the **Reshape** tool.

  Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Custom Landscape Area Tag

Creating a Custom Landscape Area Tag

In addition to providing several predefined landscape area tags, Vectorworks Landmark allows designers to create custom landscape area tags.

1. From the Tag Information pane of the Landscape Area Settings dialog box, select **Set Custom Tag** from either the **Tag Header** or **Tag Body** fields. Settings for the landscape area tag currently displaying in this field are displayed in the Set Custom Tag dialog box.

   The Set Custom Tag dialog box opens.

   If editing an existing tag through the Object Info palette, the selected tag's data displays.

2. Select values from the predefined plant record fields (including an empty string to skip the field value) in the order they should be listed, and type the delimiter text that will appear between values. A combination of up to six landscape area values and plant record fields can be specified for the **Tag Header** and **Tag Body** fields.

   The **Tag Appearance** field displays a static text preview of the custom tag. To preview a long tag in its entirety, Click-drag the bottom right corner of the dialog box to resize it.

3. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Landscape Areas

Creating a Plant Line

The **Plant Line** command creates a freehand plant line along a line, polyline, or polygon. It can be used to represent a single undefined plant, line of plants, or general plant mass. No specific plant symbol information is included in a plant line; use the **Landscape Area** tool if defined plant areas are necessary (see "Creating Landscape Areas" on page 834).

To create a plant line:

1. Select the object (line, polyline, or polygon) with the **Selection** tool.
2. Select the **Plant Line** command from the appropriate menu:
   - Designer workspace: **AEC > Plants > Plant Line**
   - Landmark workspace: **Landmark > Plant Line**

   The Plant Line dialog box opens. Suggested values are based on the selection's perimeter.

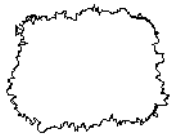
Click to show/hide the parameters.

Parameter	Description
Segment Length	Maximum length of each plant line segment; longer segments create a rougher plant line appearance
Thickness	Maximum perpendicular distance of each segment; a higher thickness value results in a rougher plant line appearance

Parameter	Description
Corner Types	Specifies the type of corner to create between segments; each type produces a different plant line effect
Delete original	Select to remove the object that forms the basis of the plant line

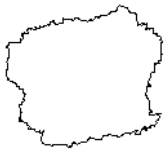
3. Select the desired plant line parameters and click **OK**. The plant line is created. If desired, apply colors, textures, and other attributes to the plant line with the Attributes palette.

Normal Corner Type
Segment Length: 0.16
Thickness: 0.0251



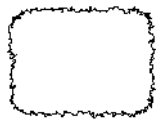
Cubic Corner Type
Segment Length: 0.0516
Thickness: 0.251

Bézier Corner Type
Segment Length: 0.16
Thickness: 0.251



Arc Corner Type
Segment Length: 0.0516
Thickness: 0.251

Random Corner Type
Segment Length: 0.0516
Thickness: 0.0251



Plant Line command executed twice

Plant Line command executed three times

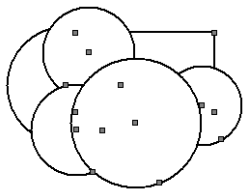
The Attributes Palette

Creating a Vegetation Line

The **Vegetation Line** command creates a vegetation line around a selection of closed objects or symbols. It can be used to represent a massed collection of undefined vegetation.

To create a vegetation line:

1. Select the closed objects or symbols that will form the basis of the vegetation line. The items should overlap.



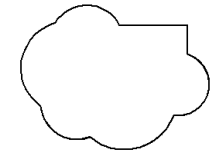
2. Select the **Vegetation Line** command from the appropriate menu:
   - Designer workspace: **AEC > Plants > Vegetation Line**
   - Landmark workspace: **Landmark > Vegetation Line**

   The Vegetation Line dialog box opens. Select the type of vegetation line to create, and whether the original objects should be deleted or retained.

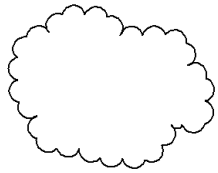
Click to show/hide the parameters.

Parameter	Description
Standard Vegetation Line	Creates an outline based on the outer perimeter of the selections
Cloud Vegetation Line	Creates a cloud outline based on the parameters specified
Minimum/Maximum Radius	Specifies the minimum and maximum radius of each arc in the cloud
Billow Height	Specifies the cloud billow height factor
Delete Original	Select to delete the original selection of objects when creating the vegetation line

3. Click **OK** to create the vegetation line. If desired, apply colors, textures, images, hatches, gradients and other attributes to the vegetation line with the Attributes palette. See “The Attributes Palette” on page 1093 for more information on applying attributes.



Standard Vegetation Line



Cloud Vegetation Line

The vegetation line can be assigned to a class (with the desired attributes). The original plant symbols can remain hidden by a vegetation line with a solid fill; however, the underlying vegetation can be revealed by hiding the vegetation line class.

Show/Hide Plant Styles

The **Show or Hide Plant Styles** command controls the visibility of the outline, massing, and shadow parameters set on the Render pane of a plant definition (see “Plant Definition: Render Pane” on page 789), on the Render pane of the plant settings (see “Plant Settings: Render Pane” on page 797), as well as on the Object Info palette.

These effects display in Top/Plan view only. In a complex drawing with many plants, these settings can take a significant amount of time to display and edit, and it is useful to toggle the effects on and off.

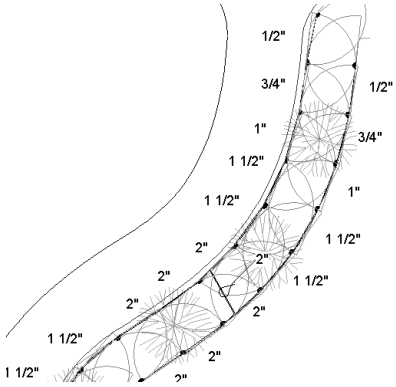
To show or hide the plant styles:

1. Select **View > Show > Show or Hide Plant Styles**.
2. If the plant styles are currently hidden, this command displays the outline, massing, and shadows of all plant objects with those parameters set.

   If the plant styles are currently visible, this command hides the effects. The **Outline** selection is saved for the plants, even when hidden.

Irrigation

The irrigation tools in the Site Planning tool set insert generic drip emitters, irrigation heads, and irrigation lines into the drawing.



A selection of standard pre-configured manufacturer's irrigation products are available through the Resource Browser, and are located in the default irrigation libraries from the [Vectorworks]\Libraries folder that is included with the Vectorworks Landmark product (see "Resource Libraries" on page 219).

For more information on the Resource Browser, see "Using the Resource Browser" on page 221.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Drip Emitter

A generic drip emitter can be inserted into the drawing with the **Drip Emitter** tool. Alternatively, use a drip emitter from one of the irrigation object libraries.

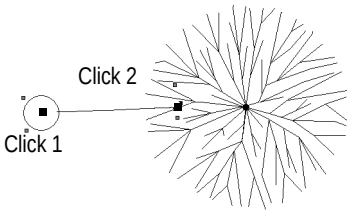
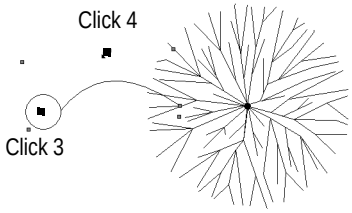
To insert a drip emitter:

1. Click the **Drip Emitter** tool from the Site Planning tool set.
2. Click in the drawing to insert the drip emitter. Click again to set the rotation.

   The first time the tool is used, the Drip Emitter Properties dialog box opens. Accept the default parameters and click **OK** to insert the drip emitter.
3. The drip emitter properties can be changed in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Manufacturer	Displays the manufacturer's name or "Generic"
Model	Displays the manufacturer's model number
Type	Options include Pressure Compensating and Non-Pressure Compensating
Head Style	Selects the type of emitter head (Micro-Spray, Drip Manifold, or Sub-surface Drip)
Spray Nozzle	For Micro-Spray head style, indicates the arc degrees of spray
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Working Pressure	Displays the water line pressure, for use in worksheets
Flow Rate	Displays the flow rate per hour in either metric or imperial units
Unit	Selects the units for the display of flow rate
Scale Factor	Determines the size of 2D drip emitter display
Microtube Control	For Sub-surface Drip head style only, displays the coordinates of microtube placement. Coordinates **1X/Y** position the control point that defines the microtube's Bézier curve; coordinates **2X/Y** position the control point that defines the microtube's end point. Alternatively to entering coordinates, you can position the control points manually. The Sub-surface drip emitter contains control points in the center of the symbol; position the first control point to determine the microtube end point, and the second to define the curvature.  Click in the center of the drip emitter to obtain the control point, and click again to place the end point  Click a third time in the center of the drip emitter to obtain the second control point, and click again to define the curvature

Inserting an Irrigation Head

A generic irrigation head can be inserted into the drawing with the **Irrigation Head** tool. Alternatively, use an irrigation head from one of the irrigation object libraries.

To insert an irrigation head:

1. Click the **Irrigation Head** tool from the Site Planning tool set.
2. Click in the drawing to insert the irrigation head. Click again to set the rotation.

   The first time the tool is used, the Irrigation Head Properties dialog box opens. Accept the default parameters and click **OK** to insert the irrigation head.
3. The irrigation head properties can be changed in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Manufacturer	Manufacturer's name or "Generic"
Model	Manufacturer's model number
Spray Pattern	Selections include Circular, Rectangular, and Special
Head Type	Indicates a Fixed Spray or Rotary sprinkler
Body	Spray head height (for use in worksheets)
Nozzle	Manufacturer's nozzle type

Parameter	Description
Spray Radius or Width	For circular spray patterns, displays the radius of the spray; for rectangular spray patterns, displays the spray width
Length of Rect Spray	Displays the length of a rectangular spray pattern
Show Spray Pattern	Select to display the irrigation spray pattern
Spray Arc	Indicates the arc of the spray pattern and determines the spray pattern display
Trajectory	Angle (in degrees) of water as it is thrown from the spray head (for use in worksheets)
Working Pressure	Water line pressure (in psi or kg/cm3) for use in worksheets
Spacing	Indicates triangular or square spacing
Precipitation Rate	Number of inches or millimeters of water placed over a certain area per hour, for use in worksheets
Head Scale Factor	Determines the size of 2D irrigation head display
Unit	Specify either Metric or Imperial units

The spray pattern can be hidden and displayed with the **Show or Hide Spray Pattern** command (see “Show/Hide Spray Pattern” on page 844).

Inserting an Irrigation Line

An irrigation line can be inserted into the drawing with the **Irrigation Line** tool. Alternatively, use an irrigation line from one of the irrigation object libraries.

To insert an irrigation line:

1. Click the **Irrigation Line** tool from the Site Planning tool set.
2. Click and drag to define the length of the irrigation line.

   The first time the tool is used, the Irrigation Line Properties dialog box opens. Accept the default parameters and click **OK** to insert the irrigation line.
3. The irrigation line properties can be changed in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Length	Displays irrigation line length
Type	Indicates the irrigation line material (PVC, PE, Galvanized Steel, or Copper)
Diameter	Displays irrigation tubing diameter
Has Sleeve	Select if the irrigation line has a sleeve
Sleeve Diameter	When the **Has Sleeve** option is selected, indicates the tubing diameter of the sleeve

Show/Hide Spray Pattern

The **Show or Hide Spray Pattern** command controls the visibility of the spray pattern of irrigation head objects.

To show or hide the spray pattern:

1. Select **View > Show > Show or Hide Spray Pattern**.
2. If the spray patterns are currently hidden, this command causes the spray pattern of all irrigation head objects to become visible. In the Object Info palette, **Show Spray Pattern** is selected automatically.

   If the spray patterns are currently visible this command causes the spray pattern of all irrigation head objects to become hidden. In the Object Info palette, **Show Spray Pattern** is automatically deselected.

Parking Tools

A basic parking tool, **Parking Spaces**, creates simple parking space graphics in any Vectorworks workspace.

Two additional parking tools available in Vectorworks Landmark, **Parking Area** and **Parking Along Path**, create parking spaces automatically within a defined area or along a path. The parking tools allow for islands, accessible spaces, head-in (angled) parking, parallel parking, and access lanes; a parking schedule counts the total number of parking spaces for each type of parking tool.

Alternatively, in Vectorworks Landmark, draw a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

Parking areas and parking along path objects can be sent to a site model surface with the **Send to Surface** command, but they do not modify the site model.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Parking Spaces

To create basic parking spaces:

1. Select the **Parking Spaces** tool from the appropriate tool set:
   - Architect and Landmark workspaces: Site Planning tool set
   - Spotlight workspace: Building Shell tool set

The **Parking Space** tool is available for Vectorworks Fundamentals software but is not present in the Fundamentals workspace. It can be added to the Fundamentals workspace (see “Creating or Editing Workspaces” on page 1835).

2. Click once to place the parking spaces object. Click again to set the object rotation.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.

3. A basic parking spaces object is placed in the drawing.

Parking space parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file’s content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See “Using Text Styles” on page 393 and “Formatting Text” on page 389.
Space Style	Select the type of parking space: • Single Bay: Draws spaces on one side of the header stripe • Double Bay: Draws spaces on both sides of the header stripe • Parallel: Creates parallel spaces (along a curb, for example) instead of head-in parking
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
No. of Standard Spaces	Specify the number of spaces to create
Space Width	Indicates the parking space width
Space Length	Indicates the parking space length (for parallel spaces, this is the length along the curb)
Space Angle	For single and double bay parking, specifies the angle of each parking space
Header Style	Select the header style: • Straight: The header stripe remains straight • Angled: The header stripe stays perpendicular to the space angle
No. of Accessible Spaces	Specifies the number of spaces with a different width applied for accessibility
Accessible Width	Sets the space width for accessible spaces
Accessible Location	Indicates whether the location of the accessible space(s) is at the beginning or end of the parking row
Show Accessible Symbol	Select to show an accessible parking symbol in each designated parking space. Vectorworks software includes an international standard accessibility symbol that can be edited by the user. 
Add Accessible Aisle Spaces	Select to add aisle spaces adjacent to accessible parking spaces
Aisle Width	Enter a value for the accessibility aisle width
Aisle Location	Select an aisle location relative to the accessible parking spaces
Aisle Fill	Click to open the Choose Fill dialog box, and select the aisle fill from options present in the drawing (see “Fill Attributes” on page 1098)
Show Header Stripe	Toggles the display of the parking header stripe
Show Start Stripe	Toggles the display of the side of the first parking space
Show End Stripe	Toggles the display of the side of the last parking space
Space Numbering	Adds a parking space number to each parking space; spaces are numbered from left to right and top to bottom
Start Numbering At	Specifies the starting number to begin parking space numbering
Show Bumpers	Toggles the display of the parking space bumpers (stops)
Show 3D Stripes	Toggles the display of the parking spaces in a 3D view

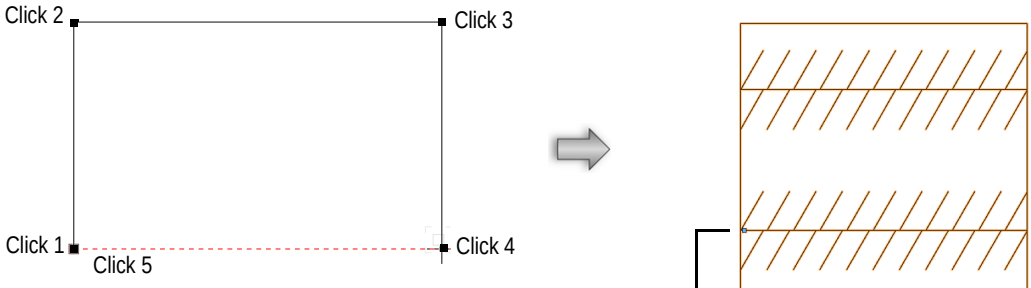
Parking Tools

Creating a Parking Area

To create a parking area:

1. Select the **Parking Area** tool from the Site Planning tool set.
2. Click to begin drawing the parking polyline; click to set each polyline vertex. Click the mouse at the start point to complete a closed polyline, or double-click the mouse to create an open polyline. For more information on polylines, see “Polyline Tool” on page 298.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.
3. The parking spaces are automatically created within the area defined by the polyline.



Parking Area tool in Corner Vertex mode

Once created, the parking area can be edited by several methods.

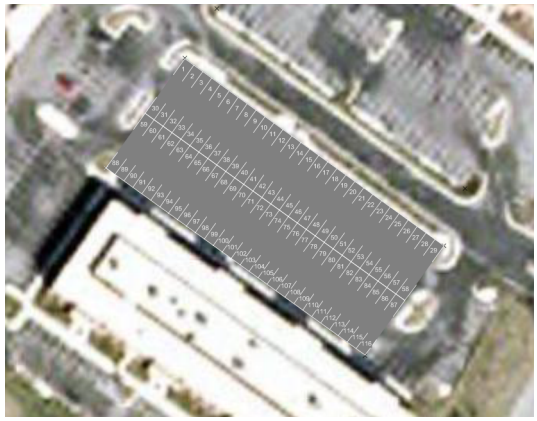
- Adjust the parking space origin by dragging the parking area control point.
- Reshape the parking area polyline with the **Reshape** tool.
- Set a solid fill color for the parking area and a solid pen color for the space numbers through the Attributes palette.
- Change the parking space numbering font, size, and style with the **Text** menu, or assign a text style.
- The parking area displays in a 3D view; if desired, assign a texture to the parking area with the Render tab of the Object Info palette (Renderworks required).

Parking area parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file’s content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See “Using Text Styles” on page 393 and “Formatting Text” on page 389.
Lane Width	Specifies the width of the lane between rows of parking spaces
Row Angle	Sets the angle for the parking header stripe
Space Width	Indicates the width of each parking space
Space Length	Indicates the length of each parking space
Space Angle	Specifies the angle of each parking space

Parameter	Description
Show Header Stripe	Toggles the display of the parking header stripe
Show Perimeter	Toggles the display of the original polyline
Space Numbering	Adds a parking space number to each parking space; spaces are numbered from left to right and top to bottom
Start Num At	Specifies the starting number to begin parking space numbering
Num Spaces	Displays the number of parking spaces created



Aerial parking lot photo with parking area superimposed

Parking Tools
Creating Parking Islands

Creating Parking Islands

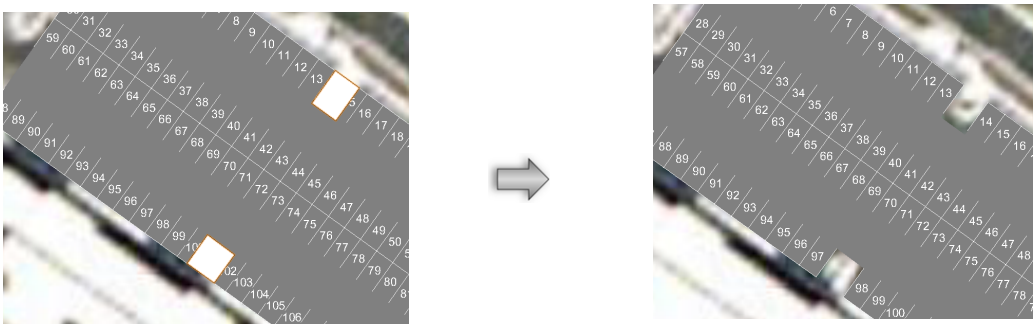
Many parking areas include islands where trees are planted or parking spaces are otherwise separated. By using the **Clip Surface** command, the parking area can be adjusted to account for parking islands, or for accessible parking. The parking spaces automatically adjust for the presence of the parking islands.

To create parking islands within a parking area:

1. Draw the 2D clipping objects on the parking area.
2. Select the clipping objects and the parking area. See “Clip Surface” on page 1083 for more information.
3. Select **Modify > Clip Surface**.

   The parking area is clipped, removing the areas under the clipping objects.
4. Delete the clipping objects.

   The parking area automatically adjusts for the removed spaces, renumbering the parking spaces.



Although the **Parking Space** tool offers several parameters for easily defining accessible parking spaces, the **Clip Surface** technique can also be used to create accessible spaces. Do not remove the clipping object after clipping; apply the same fill color and then apply an accessible parking graphic image to the area. Accessible spaces created this way are not counted in parking space reports.

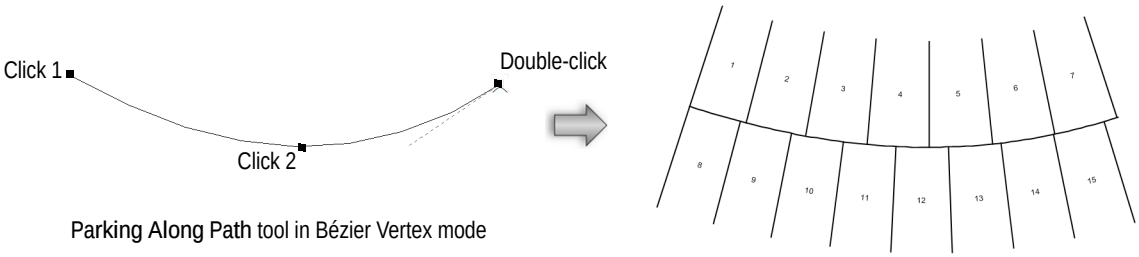
Parking Tools

Creating Parking Along a Path

To create parking along a path:

1. Select the **Parking Along Path** tool from the Site Planning tool set.
2. Click to begin drawing the parking polyline path; click to set each polyline vertex. Double-click the mouse to create an open path polyline, or click at the start point to complete a closed polyline. For more information on polylines, see “Polyline Tool” on page 298.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.
3. The parking spaces are automatically created along the polyline path.



Parking Along Path tool in Bézier Vertex mode

Once created, the parking along path object can be edited by selecting the parking object; then select **Modify > Edit Parking Along Path** and reshape the parking area polyline with the **Reshape** tool; click **Exit Path** to return to the drawing.

Parking along path parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Space Width	Indicates the width of each parking space. Spaces along a curve may widen along the length of the space; the space width is defined as the minimum width between the two sides.
Space Length	Indicates the length of each parking space
Space Numbering	Adds a parking space number to each parking space; spaces are numbered from left to right and top to bottom
Start Num At	Specifies the starting number to begin parking space numbering
Num Spaces	Displays the number of parking spaces created
Spaces on Left of Poly	Includes spaces along the left side of the parking path
Spaces on Right of Poly	Includes spaces along the right side of the parking path

Aerial parking lot photo with curved parking along path superimposed

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parking Tools

Creating Parking Reports

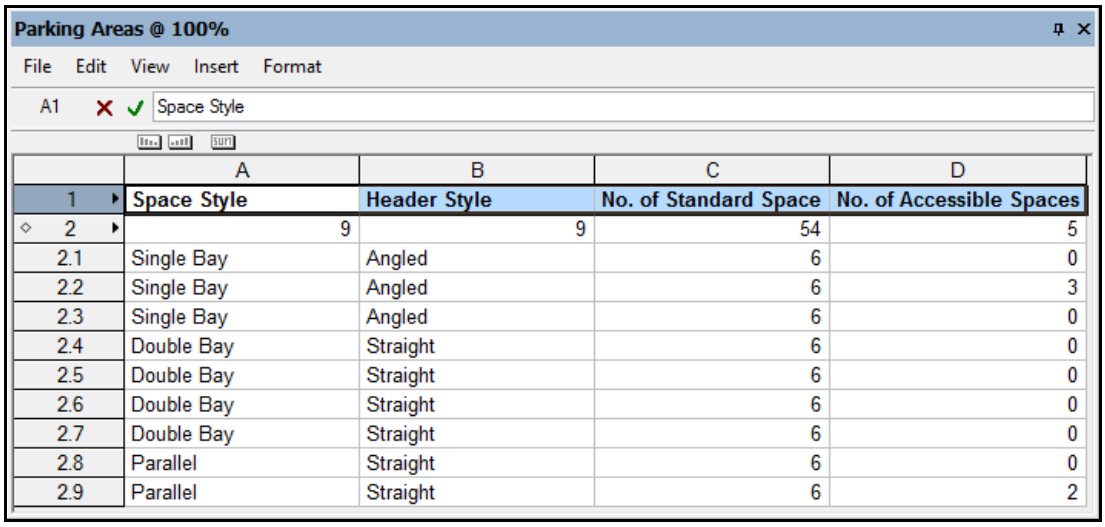
Parking report worksheets can be created for each type of parking tool: parking areas, parking along path objects, and parking spaces. If using a combination of parking types, a report may be needed for each type.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To create a parking report schedule:

1. Select **Tools > Reports > Create Report**.
2. Select the type of parking from the **Listing objects with record** list, and select the columns to include in the worksheet.

   See “Creating Reports” on page 1316 for more information.
3. Click **OK** to create the report.

   The worksheet calculates the total number of spaces for all parking objects of the same type.



	A	B	C	D
1	Space Style	Header Style	No. of Standard Space	No. of Accessible Spaces
2	9	9	54	5
2.1	Single Bay	Angled	6	0
2.2	Single Bay	Angled	6	3
2.3	Single Bay	Angled	6	0
2.4	Double Bay	Straight	6	0
2.5	Double Bay	Straight	6	0
2.6	Double Bay	Straight	6	0
2.7	Double Bay	Straight	6	0
2.8	Parallel	Straight	6	0
2.9	Parallel	Straight	6	2

Parking Tools

Lighting Design

Lighting Design Workflow

The Vectorworks Spotlight product provides the lighting designer with all the tools necessary to plan and create a light plot and its associated paperwork. The following general workflow is recommended.

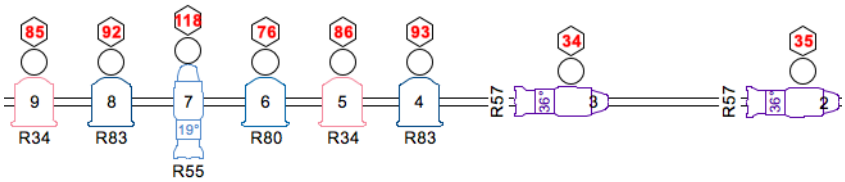
1. Create a template file that you can use to start new design files, or from which you can import standard elements into other files. The template can contain all of the resources, classes, and design layers you use on a regular basis. You might also set options you commonly use, such as the default object attributes and page size. It is recommended that you plan to place the stage, focus points, and scenic elements on separate design layers; plan to place the lighting positions, lighting instruments, and accessories together on one layer. All design layers should use the same scale. Select the **File > Save As Template** command to save the file.
2. Create a new project file from your template file.
3. Select the **File > Document Settings > Spotlight Preferences** command to set default lighting device parameters, and to specify whether universe assignment should be handled automatically. If you use Lightwright, set up automatic data exchange; this creates a file with the same name as the Vectorworks file with an .xml file extension, which you can then open in Lightwright. Changes that you make in both the Lightwright and Vectorworks programs are shared seamlessly.
4. Copy or import design elements (such as theater architecture or scenic designs) from other files, as needed. From a Vectorworks file, you can paste or import items directly into your file (you might want to apply your own classes to them first). Alternatively, create a design layer viewport to reference an external Vectorworks file; your file will be updated automatically if the external file changes, and the external classes will not be imported into your file. The Vectorworks program can also import drawing elements from several file types, such as DXF/DWG, PDF, and SketchUp.
5. Use the **Lighting Pipe**, **Lighting Pipe Ladder**, **Straight Truss**, and **Curved Truss** tools to create lighting position geometry. You can also draw your own 2D or 2D/3D geometry. Select the **Spotlight > Object Conversion > Convert to Light Position** command to convert each object into a light position. From the Object Info palette, enter a **Z** value to indicate the height of each position.
6. To use the same geometry for multiple lighting positions, select the option to create a symbol when you convert the geometry. Make the symbol active in the Resource Browser, and then use the **Lighting Position** tool to insert additional lighting positions.
7. Use the **Focus Point** tool to create focus areas. In step 10, you will assign these focus points to specific lighting instruments.
8. Select the **Label Legend Manager** command to create label legends for your lighting instruments. The label legend specifies which data fields to display for the instruments, as well as how the fields are laid out.
9. Select the **Lighting Instrument** tool to place lighting instruments on the lighting positions. Thousands of instrument symbols are available for import, from resource libraries provided with the Vectorworks application, from subscription libraries available through Vectorworks Service Select, and from third-party providers.

   The lighting instruments in the Vectorworks Spotlight product represent more than simply a graphical symbol. An instrument contains design information, such as labels, focus, position, channel, and color. Instruments are labeled with lighting information according to the label legend format. Use the standard instrument symbols, or convert your own custom symbols to instruments.

   You can also draw your own instrument symbols as needed. The position and instrument height are filled in automatically according to the lighting position.

   From the Object Info palette, specify the instrument purpose, color, and dimmer.
10. Select the **Spotlight > Focus Instruments** command to specify a focus point for each instrument.

11. Use the **Lighting Accessory** tool to add accessories to lighting instruments. As with lighting instruments, a wide variety of accessory symbols are available for import, or you can draw your own.
12. Fine-tune the lighting design using the following tools and commands:
    - Select the **Draw Beam** option in the Object Info palette and check the light beam and field for each lighting instrument.
    - Use the **Photometer** and **Photometric Grid** tools to check the footcandles present at specific spots on the stage.
    - Use the **Align and Distribute Items** tool to more precisely align instruments to their positions.
    - Select the **Spotlight > Visualization > Create Plot and Model View** command to automatically create a 3D model for a selected vertical or angled lighting position. This creates a design layer viewport on a model layer, which can be rotated to show the lighting position accurately in sections and elevations.
13. Use the **Soft Goods** tool to insert curtains, borders, scrims, and pipe-and-drape assemblies where needed.
14. Generate paperwork for yourself and others using one or more of the following commands:
    - Select the **Spotlight > Reports > Generate Paperwork** command to create an instrument schedule and other typical electrician paperwork from within the Vectorworks program.
    - Select the **Spotlight > Reports > Create Report** command to create a database worksheet, which allows you to edit lighting instrument data in a spreadsheet format from within the Vectorworks program.
    - If you use the Lightwright automatic data exchange feature, your exchange file is ready to share when needed.
15. Use the **Instrument Summary** tool to create a summary graphic of the instruments and accessories that are used in the light plot. To create a summary graphic for a selected lighting position, click **Insert Position Summary** in the Object Info palette. To compare the required totals to current inventory, select the **Spotlight > Reports > Lighting Inventory Setup** command to enter inventory information.
16. Use commands on the **View** menu (**Create Viewport**, **Create Detail Viewport**, **Create Section Viewport**) to create the viewports you want to print or export, such as the main light plot, detail views, section views, and hanging cards. Show or hide classes and layers, and add title blocks, borders, dimensions, and notes as needed.
17. Print or export the sheet layers, as needed. The **File > Publish** command allows you to easily publish selected sheet layers to a printer, or to PDF, DXF/DWG, DWF, and image files. You can also save the publish settings and use them again later. The **File > Export** menu has options for exporting to several additional file types.



Spotlight Terminology

In Vectorworks, you use the **Lighting Instrument** tool to place a lighting instrument on the drawing. Optionally, you can use the **Lighting Accessory** tool to place a lighting accessory on the drawing, and associate it with the appropriate instrument. Because they are so closely linked, all instrument and accessory objects are referred to as “lighting devices” when they are edited in the Object Info palette, Properties dialog box, or Lighting Device dialog box. The **Device Type** field indicates whether an accessory or instrument is being edited. A lighting accessory has a type of Accessory or Static Accessory; any other **Device Type** indicates a lighting instrument.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Light Plot Structure

A structured system of layers and classes makes selecting, viewing, and printing items considerably easier. When creating a light plot, develop a standard layer and class structure, along with a system for assigning items to the appropriate layers and classes. Some objects include the ability to be automatically assigned to classes.

For more information on layers and classes, see “Layers” on page 161 and “Classes” on page 176.

We recommend that you place the stage, focus points, and scenic elements in separate design layers; place the lighting positions, lighting instruments, and accessories together on one layer. Select **Tools > Organization** and click the Design Layers tab to create and manage layers.

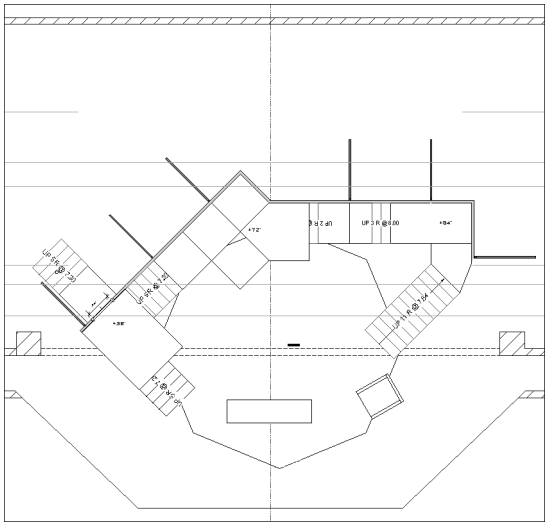
Use the same scale for all the design layers.

Place light plot items in their own classes. This allows you to globally turn off the display of certain classes for viewing and printing. Select **Tools > Organization** and click the Classes tab to create and manage classes. Lighting devices can be automatically assigned to classes determined at setup; see “Lighting Device Setup” on page 95.

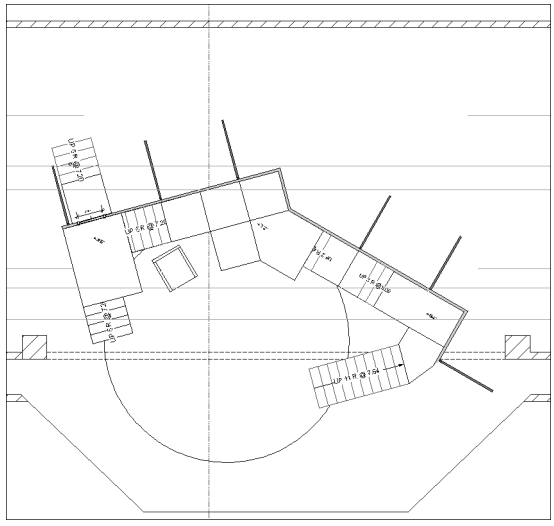
Vectorworks automatically creates certain classes to control instrument label visibility. These include the Label class and its subclasses, and the Setup Notes class.

Layer and Class Examples

An example of the use of layers and classes would be a show file with all of the scenery on a single design layer, and classes created to represent Act 1 and Act 2. When adding scenic elements, place them on the scenery design layer, and assign them to either the Act 1 or Act 2 class. With this structure, use layer visibility to view only the scenery design layer, and class visibility to turn the Act 1 or Act 2 classes on and off. This method allows you to view the position of the set pieces in each act independently.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Scenery position for Act 1

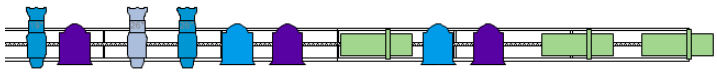


Scenery position for Act 2

Another way to take advantage of structuring is to also use classes for the lighting devices. In a situation with multiple shows in repertory, assign each device to the class of the show where it is used. This allows you to display each show's devices independently of the other shows.

Creating Lighting Positions

Lighting positions in represent the hanging points for instruments; they should be placed on the light plot before the instruments are added. Lighting positions manage the numbering of instruments according to the instruments' location on the lighting position.



Defining the lighting positions first is useful because once the space has been created, with the stage and lighting positions defined, the file can be used as a template for future light plots in that space. For more information on creating templates, see "Creating Templates" on page 75.

> Place the lighting positions in their own design layer and in their own classes. A drawing structure based on layers and classes facilitates selecting, viewing, and printing the light plot.

Creating a lighting position is generally a two-step process. First, insert or draw the geometry of the lighting position. Then, convert the geometry into an intelligent lighting position object, ready for instrument placement.

Several different types of objects can be used as the basis for a lighting position object:

- Lighting pipes
- Lighting pipe ladders
- Straight trusses
- Curved trusses
- Custom 2D or 2D/3D geometry that you draw

To convert multiple objects to a single lighting position, group the objects first. Once the geometry is complete, convert it into an intelligent lighting position object with the **Convert to Light Position** command. You have the option to create a symbol from the geometry, or to embed the geometry into the object. If you choose to create a symbol, you can use it to insert multiple positions with the **Lighting Position** tool. If this is a uniquely shaped lighting position, or if you want to be able to edit the geometry more easily, choose instead to keep the geometry.

Use the **Lighting Position** tool to insert symbols from the current file, the symbol libraries, or a Favorites file.

Placing Lighting Pipe Objects

Insert and configure a lighting pipe object with the **Lighting Pipe** tool.

 To insert a lighting pipe object:

1. Click the **Lighting Pipe** tool from the Spotlight tool set.

   Alternatively, create a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

2. Click in the drawing area to begin drawing the lighting pipe.

   If this is the first time a lighting pipe has been placed on the drawing, the Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**.

3. To draw a multi-segment pipe, click to set the end of one segment and start the next.

   Create a curved pipe by changing the vertex type (see “Creating Polylines” on page 298).

4. Double-click to complete the lighting pipe polyline.

The lighting pipe parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Total Pipe Length	Specifies the total length of the lighting pipe; for a single-segment lighting pipe, the length can be edited by changing this value. (Change the segment length or edit multi-segment pipes with the **Reshape** tool.)
Arc Radius	For single-segment curved pipes, specifies the radius of the arc
Position Name	Enter the name to be used for the symbol when converting the pipe to a lighting position
Location	Optionally, specifies the location for use in reports
Pipe	
Draw Pipe	Shows or hides the 2D component of the lighting pipe
Line Type	Select either a single or double line for Top/Plan views
Diameter	Sets the pipe diameter for all views
End Marker	Specifies whether to display the end(s) of the pipe with an end marker in Top/Plan view
Tick Mark	

Parameter	Description
Draw Tick Marks	Creates evenly-spaced marks to assist with lighting instrument placement
Draw Loci	Adds a 2D and 3D locus at the tick mark positions for visibility in 3D views
Centers	Specifies the distance between tick mark centers
Centered on	Spaces the tick marks based on the total length of the pipe, or based on each segment of a pipe that is considered divided into two equal sections
Origin	Starts the tick mark placement at the center of the pipe or split equally from the center of the pipe  On Center Split Center
Type	Select Line or Dot as the 2D tick mark indicator
Size	Specifies the length of line tick marks or the diameter of dot tick marks
Gap Length	Adds a gap on either side of the tick mark; specify the gap length
Offset	Specify a positive or negative value to shift the tick marks right or left from their centers
Align Tick Marks to Grid	Aligns the center tick mark with the snap grid (see “Grid Snapping” on page 133); this does not change the tick mark spacing but it may change the **Offset** value
Footprint/3D	
Show Footprint	Creates a 2D projection of the 3D component in Top/Plan view; this is useful if the lighting pipe is offset or rotated
Align 3D with Footprint	When selected, aligns the 3D component and the footprint in 3D views. This is the default behavior for the hybrid object to appear correctly in both 2D and 3D views. Deselecting this option and setting offset and rotation values is useful only for workflows where a hybrid appearance is not critical; however, lights placed in one view may not appear positioned correctly in all views.
Origin Offset X/Y	Offsets the 3D component from its origin along the X/Y axes
Rotation	Rotates the 3D component about its origin
I Rotation	Rotates the 3D component along the I axis (parallel to the width)
J Rotation	Rotates the 3D component along the J axis (perpendicular to the lighting pipe); a positive value pivots the pipe up, away from the ground
Classes	
Auto-Class	Uses the class specified in **Settings** as the root class for the lighting pipe components; this root class is automatically set as the prefix for the sub-classes
Pipe	Select a class for the lighting pipe to control its fill color and/or texture and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Lighting Pipe Class> which places the pipe in the same class as the lighting pipe object (specified in **Settings**).

Parameter	Description
Tick Marks	Select a class for the tick marks to control their fill color and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Lighting Pipe Class> which places the tick marks in the same class as the lighting pipe object (specified in **Settings**).
End Marker	Select a class for the end markers to control their fill color and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Lighting Pipe Class> which places the end markers in the same class as the lighting pipe object (specified in **Settings**).
Footprint	Select a class for the footprint to control its fill color and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Lighting Pipe Class> which places the footprint in the same class as the lighting pipe object (specified in **Settings**).
Settings	Opens the Lighting Pipe Settings dialog box, to set the default class for the lighting pipe. Select the **Make default for new documents** option to always use the specified class as the default class for lighting pipe objects in new files.
Vertex Parameters	Edits the pipe vertices; see “Editing Vertex-Based Objects” on page 1002

To use the lighting pipe as a lighting position, convert it to a lighting position object; see “Creating a Lighting Position Object” on page 865.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Lighting Positions
Creating Lighting Positions

Placing Lighting Pipe Ladders

Insert and configure a lighting pipe ladder with the **Lighting Pipe Ladder** tool.

To insert a lighting pipe ladder:

1. Click the **Lighting Pipe Ladder** tool from the Spotlight tool set.
2. Click in the drawing area to insert the ladder. The ladder is a rectangular object, and can be inserted in Center-line Placement mode or Edge Placement mode.

Placement Mode	Description
Center-line	Click once, and then again, to define the width through the center of the ladder. Click again to specify the width of half the ladder.
Edge	Click once, and then again, to define the length along the edge of the ladder. Click again to specify the grid ladder width.

If this is the first time a lighting pipe ladder has been placed on the drawing, the Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The lighting pipe ladder parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Ladder Width/ Height	Specifies the lighting pipe ladder dimensions
Guides	Select how to draw the ladder based on the preview guide rectangle: on the ladder center, inside, or outside
Number of Rungs	Specifies the number of ladder rungs
Evenly Spaced Rungs	Spaces the rungs evenly within the ladder frame; deselect to specify custom rung spacing
Custom Rung Spacing	Opens the Custom Rung Spacing dialog box, to specify the spacing between each ladder rung. Enter the distance between the rungs; the first and final distances are automatically calculated. Click **Calculate Even Spacing** to automatically space the rungs evenly, as a starting point for spacing or to reset the values.
Position Name	Enter the name to be used for the symbol when converting the ladder to a lighting position
Location	Optionally, specifies the location for use in reports
Pipe	
Draw Pipe	Shows or hides the 2D component of the lighting pipe ladder
Line Type	Select either a single or double line for Top/Plan views
Diameter	Sets the pipe diameter for all views
Tick Mark	
Draw Tick Marks	Creates evenly-spaced marks to assist with lighting instrument placement
Draw Loci	Adds a 2D and 3D locus at the tick mark positions for visibility in 3D views
Centers	Specifies the distance between tick mark centers
Origin	Starts the tick mark placement at the center of one side of the ladder or split equally from the center of one of the ladder sides  On Center Split Center
Type	Select Line or Dot as the 2D tick mark indicator
Size	Specifies the length of line tick marks or the diameter of dot tick marks
Gap Length	Adds a gap on either side of the tick mark; specify the gap length
Offset	Specify a positive or negative value to shift the tick marks right or left from their centers
Align Tick Marks to Grid	Aligns the center tick mark with the snap grid (see “Grid Snapping” on page 133); this does not change the tick mark spacing but it may change the **Offset** value
Footprint/3D	

Parameter	Description
Show Footprint	Creates a 2D projection of the 3D component in Top/Plan view; this is useful if the lighting pipe ladder is offset or rotated
Align 3D with Footprint	When selected, aligns the 3D component and the footprint in 3D views. This is the default behavior for the hybrid object to appear correctly in both 2D and 3D views. Deselecting this option and setting offset and rotation values is useful only for workflows where a hybrid appearance is not critical; however, lights placed in one view may not appear positioned correctly in all views.
Origin Offset X/Y	Offsets the 3D component from its origin along the X/Y axes
Rotation	Rotates the 3D component about its origin
I Rotation	Rotates the 3D component along the I axis (parallel to the width)
J Rotation	Rotates the 3D component along the J axis (perpendicular to the lighting pipe ladder); a positive value pivots the ladder up, away from the ground
Classes	
Auto-Class	Uses the class specified in **Settings** as the root class for the lighting pipe ladder components; this root class is automatically set as the prefix for the sub-classes
Pipe	Select a class for the lighting pipe ladder to control its fill color and/or texture and visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Lighting Pipe Ladder Class> which places the pipe in the same class as the lighting pipe ladder object (specified in **Settings**).
Tick Marks	Select a class for the tick marks to control their fill color and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Lighting Pipe Ladder Class> which places the tick marks in the same class as the lighting pipe ladder object (specified in **Settings**).
Footprint	Select a class for the footprint to control its fill color and/or visibility. The classes present in the drawing are listed; alternatively, create a new class by selecting New, select the default class name, or select the class named <Lighting Pipe Ladder Class> which places the footprint in the same class as the lighting pipe ladder object (specified in **Settings**).
Settings	Opens the Lighting Pipe Ladder Settings dialog box, to set the default class for the lighting pipe ladder. Select the **Make default for new documents** option to always use the specified class as the default class for lighting pipe ladder objects in new files

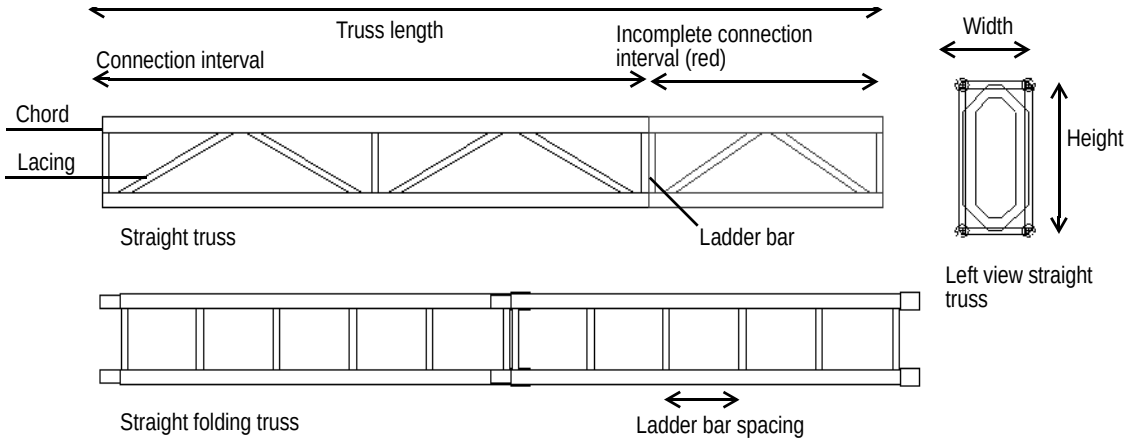
To use the lighting pipe ladder as a lighting position, convert it to a lighting position object; see “Creating a Lighting Position Object” on page 865.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Lighting Positions
Creating Lighting Positions

Inserting a Straight Truss

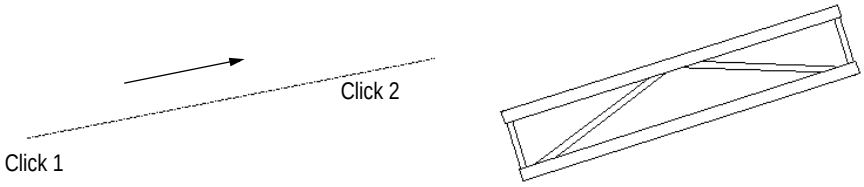
Straight truss parameters are illustrated by the following diagram.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



To add a straight truss to the light plot:

1. Click the **Straight Truss** tool from the appropriate tool set.
   - Spotlight workspace: Spotlight tool set
   - Designer workspace: Detailing tool set
2. Click in the drawing area where the truss will be located and draw a line to indicate the length of the truss. If this is the first time a straight truss has been placed on the drawing, the Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**.

   The straight truss properties can be edited later in the Object Info palette.



Trusses are 2D/3D hybrid objects. Complex trusses can increase the time required to render the model in 3D.

Click to show/hide the parameters.

Parameter	Description
Length	Specifies the length of the truss in drawing units
Profile	Select the truss profile from the list
Connection	Select the type of connection from the list
Connection Interval	Indicates the distance for dividing the truss into component sections
Folding	Select to indicate a folding truss
Pre Rigged	Select whether to include a light bar
Height/Width	Shows height and width values for the truss
Chord Profile	Select round or square chord tubing
Chord Width	Shows the diameter of round chords or the width of square chords
Ladder Bar Spacing	Specifies the interval of the ladder bars

Parameter	Description
Ladder Bar Profile	Select round or square ladder bar tubing
Top/Side Ladder Bar Diameter	Indicates the diameter of the top and side ladder bar tubing; enter 0 to draw no ladder bar
Lacing Profile	Select round or square lacing tubing
Top/Side Lacing Diameter	Indicates the diameter of the top and side lacing tubing. Enter 0 to draw no lacing.
Hanging Angle	Indicates the angle of the truss between the stage or floor and the hanging point
Rotation	Specifies the rotation about the truss axis
Show 3D Detail	Select to render the truss with greater 3D detail
Highlight	Displays odd sized truss divisions in red
Message	Provides information about the truss placement and indicates successful placement

A connection interval shorter than the defined interval length is displayed in red when **Highlight** is selected.

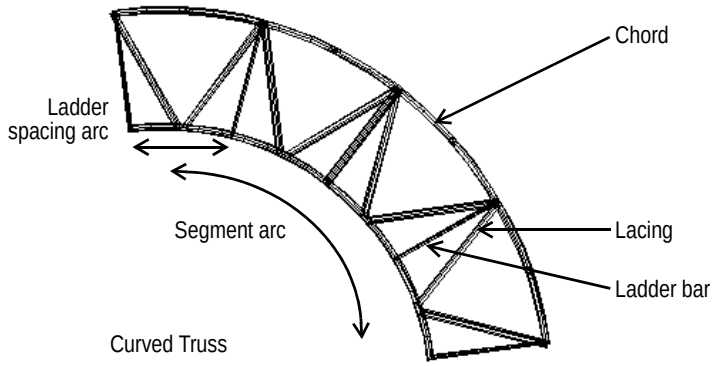
To use the truss as a lighting position, convert it to a lighting position object; see “Creating a Lighting Position Object” on page 865.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Curved Truss
Truss Configuration Requirements
Inserting Lighting Positions
Creating Lighting Positions

Inserting a Curved Truss

Curved truss parameters are illustrated by the following diagram.



To add a curved truss to the light plot:

1. Click the **Curved Truss** tool from the appropriate tool set:
   - Spotlight workspace: Spotlight tool set.
   - Designer workspace: Detailing tool set
2. Click to define the truss insertion point. Click again to set the rotation of the curved truss. If this is the first time a curved truss has been placed on the drawing, the Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The curved truss properties can be edited later in the Object Info palette.

Trusses are 2D/3D hybrid objects. Complex trusses can increase the time required to render the model in 3D.

Click to show/hide the parameters.

Parameter	Description
Radius	Specifies the radius of the truss measured from the truss center line
Total Arc	Indicates the total angle covered by the truss in degrees
Segment Arc	Specifies the interval in degrees for dividing the truss into component sections
Connection	Select the type of connection from the list
Profile	Select the truss profile from the list
Height/Width	Shows height and width values for the truss
Chord Profile	Select round or square chord tubing
Chord Width	Specifies the diameter of round chords or the width of square chords
Ladder Spacing Arc	Specifies the interval of the ladder bars in degrees
Ladder Profile	Select round or square ladder bar tubing
Top/Side Ladder Bar Diameter	Specifies the diameter of the top and side ladder bar tubing. Enter 0 to draw no ladder bar.
Lacing Profile	Select round or square lacing tubing
Top/Side Lacing Diameter	Specifies the diameter of the top and side lacing tubing. Enter 0 to draw no lacing.
Show 3D Detail	Select to render the truss with greater 3D detail
Highlight	Displays odd sized truss divisions in red
Message	Provides information about the truss placement and indicates successful placement
Draw 3D only	Select to display only the 3D component of the truss

If the final segment of a truss is shorter than the defined interval length, it is displayed in red when **Highlight** is selected.

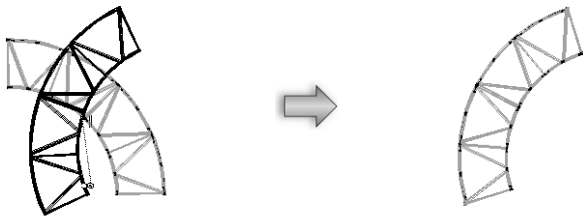
To use the truss as a lighting position, convert it to a lighting position object; see “Creating a Lighting Position Object” on page 865.

Rotating a Curved Truss

A curved truss can be rotated with the **Rotate** tool.

To rotate a curved truss:

1. Select the truss. In the Object Info palette, select **Draw 3D Only**.
2. Click the **Rotate** tool from the Basic palette.
3. Click to set a fulcrum line; click to end. Move the cursor to rotate the truss, and then click to set the rotation.



Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Straight Truss
Truss Configuration Requirements
Creating a Lighting Position Object
Inserting Lighting Positions
Creating Lighting Positions

Truss Configuration Requirements

A great deal of flexibility is incorporated in the Spotlight truss object. However, in order to create a valid truss, the wide variety of configurations leads to certain restrictions in the application of truss properties.

If a truss is drawn that does not meet these requirements, an alert message is generated.

- The width and height of the truss must be greater than three times the chord width
- The truss hanging angle must be between -90° and +90°
- A triangular truss must form a valid triangle
- The chord diameter must be greater than zero
- A pre-rigged truss cannot have lacing
- A pre-rigged truss cannot be folding
- Only box trusses can be pre-rigged
- A folding truss cannot have top lacing
- Folding is not allowed on a stacking truss
- Top lacing is not allowed on a stacking truss
- The top brace diameter must be less than or equal to the chord width
- The side brace diameter must be less than or equal to the chord width
- The connection interval must be greater than the ladder bar spacing
- Ladder bar spacing must be greater than three times the top brace size
- If the section is too small, lacing may not be drawn

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Straight Truss
Inserting a Curved Truss

Creating a Lighting Position Object

As described in “Creating Lighting Positions” on page 856, you can create lighting position geometry by placing a Spotlight lighting pipe, ladder, or truss object. You can also create your own geometry for conversion. At a minimum, a lighting position object requires at least a 2D component; it also requires a 3D component to display correctly in 3D views. If a 2D only or 3D only object is selected when you execute the command, you will be prompted to confirm that you wish to proceed with the conversion.

When you have placed the geometry in the drawing, convert it to a lighting position object. Choose whether to create a symbol from the geometry, or to simply embed the geometry in the object. If you embed the geometry, it is easier to edit later. If you create a symbol, it is available in the Resource Browser, where you can easily insert it into the drawing again with the **Lighting Position** tool, or reference or export it to other files. For more information about symbols, see “Symbols” on page 237.

To convert lighting position geometry to a lighting position object:

1. To convert multiple objects to a single lighting position, select **Modify > Group** and group the objects first.
2. Select the geometry.
3. Select **Spotlight > Object Conversion > Convert to Light Position**.
4. The Enter Text dialog box opens; specify the name of the lighting position, and then click **OK**.
5. An alert dialog box opens; click the desired option.
   - **Create Symbol** converts the selected object to a lighting position symbol that remains inserted in the drawing. (If multiple items were selected for conversion, each object is converted one at a time.) The symbol is automatically named using the lighting position name followed by -Sym (as in FOH-Sym), and it becomes the active symbol in the Resource Browser.
   - **Use Geometry** converts the selected object to a lighting position object. (If multiple items were selected for conversion, each object is converted one at a time.)
6. If you created a symbol, you can use the **Lighting Position** tool to insert additional lighting positions with the same symbol, if required. See “Inserting Lighting Positions” on page 866.

Once a lighting position is created, lighting instruments can be placed on it. See “Adding Lighting Instruments” on page 873.

Editing Lighting Position Geometry

If the lighting position object is not a symbol, you can edit it from the Object Info palette.

To edit lighting position geometry:

1. Select the lighting position object, and click **Edit Position Geometry** from the Object Info palette. Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object, and select **Edit** from the context menu.
2. The editing window opens, containing the object to be edited. A colored border around the drawing window indicates that you are in an editing mode. Edit the object as needed, and then click the **Profile Exit** button in the top right corner of the drawing window to update the object and return to the normal drawing mode.

To replace the lighting position geometry with an existing symbol:

Select the lighting position object, and enter the name of the symbol in the **Symbol Name** field in the Object Info palette.

Creating Lighting Positions

Inserting Lighting Positions

Use the **Lighting Position** tool to insert a lighting position symbol from the Resource Browser. This can be a symbol that you have created, or it can be one of the symbols imported from the libraries included with the Vectorworks Spotlight software ([Vectorworks]\Libraries); see “Resource Libraries” on page 219.

To insert a lighting position symbol:

1. Click the **Lighting Position** tool from the Spotlight tool set.
2. Select the desired lighting position symbol from the Resource Browser and choose **Resources > Make Active** to make it the active symbol definition. Alternatively, double-click on the symbol to activate it.
3. Click in the drawing to insert the active lighting position symbol definition, and then click again to set the lighting position rotation.

   If you insert the lighting position symbol without first clicking on the **Lighting Position** tool, the symbol is inserted by the **Symbol Insertion** tool. However, the lighting position functionality will not be present. When correctly inserted, the Object Info palette displays “Lighting Position” for the selected lighting position.

   Once the **Lighting Position** tool has been selected, it remains selected by default so that lighting positions can be repeatedly inserted. To place non-lighting position symbols after placing lighting positions, switch to the **Symbol Insertion** tool.
4. A different type of lighting position can be placed by changing the active symbol definition and continuing to use the **Lighting Position** tool.

The lighting position object parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Z (Height)	For 3D lighting positions, specifies the height
Rotation	Specifies the rotation angle of the lighting position
Text Style	Select a text style from either the default content or the current file’s content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See “Using Text Styles” on page 393 and “Formatting Text” on page 389.
Position Name	Specifies the name of lighting position symbol
Location	Optionally, specifies the location for use in reports
Show Name	Displays the **Position Name** on the drawing
Insert Position Summary	Inserts an instrument summary, filtered for the current lighting position; see “Creating an Instrument Summary for a Lighting Position” on page 966
Starting Number	Specifies the starting number for instrument auto-numbering
Increment	Specifies the increment for auto-numbering instruments
Numbering Direction	Specifies the page relative numbering direction
Multi-Circuit Numbering	Specifies the multi-circuit numbering format for multi-circuit instruments: • **AlphaNum**: Numbers the multicircuit instruments with A1, A2, A3... • **Num.Alpha**: Numbers the multicircuit instruments with 1.A, 1.B, 1.C... • **Num.Num**: Numbers the multicircuit instruments with 1.1, 1.2, 1.3...
Auto Number	Select to automatically number instruments and multi-circuit instruments on the lighting position

Parameter	Description
Symbol Name	If the lighting position object is symbol-based, displays the name of the symbol used to create the lighting position. If the lighting position object is not symbol-based, enter the name of an existing symbol to replace the current geometry with the symbol.
Edit Position Geometry	When the lighting position is not symbol-based, edits the geometry of the lighting position object; see “Editing Lighting Position Geometry” on page 866

The Z value of a lighting position can only be specified for lighting positions with 3D geometry. However, the Z value can be specified for the lighting instruments placed on any type of lighting position (2D, 3D, or hybrid). A lighting position created from a symbol or plug-in object automatically assumes the Z value of the original object.

If a lighting position is copied, it is automatically renamed; any associated duplicated lighting instruments are associated with the new lighting position. Deleting a lighting position does not delete the associated lighting instruments.

If instruments are moved to a different lighting position, the lighting position name is automatically updated for the instruments and the Z-height of the instruments is calculated automatically based on the lighting position Z-height.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Instrument Numbering
Creating Lighting Positions

Setting Up Instrument Label Legends

Label legends specify the labels and label formatting for lighting instruments on the light plot. Multiple label legends can be defined and then selectively applied to different instruments. The active label legend is applied to instruments as they are placed, or a label legend can be selected for existing instruments from the Object Info palette. Label legends are 2D screen plane objects. (See “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152).

Click here for a video tip about this topic (Internet access required).

Label Classes

Each label is automatically placed in its own Label class. This allows the labels to be globally turned on and off. For example, if the designer is printing the light plot for an electrician, the “Label-Purpose” class of labels can be hidden so that only electrical information labels are shown.

The labels display the lighting instrument object information. The instrument Object Info palette also shows this information (see “Lighting Instrument Properties” on page 877).

Using the Label Legend Manager

The Label Legend Manager controls the setup and modification of the instrument label styles. Select **Spotlight > Label Legend > Label Legend Manager** to set up the legends for the light plot. The Label Legend Manager dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Active	To activate the default legend for lighting instruments, select the label legend and click in the **Active** column. The active legend is indicated with a check mark. Any new instruments created will use the active legend as the default label legend; the legend name is displayed in the instrument Object Info palette, and can be changed at any time (see “Changing the Instrument Label Legend” on page 885).
Legend Name	Lists the file’s current legends
Add	Creates a new label legend and specifies the labels to include, as well as their attributes
Remove	Deletes the selected label legend; the active label legend cannot be removed
Edit Fields	Edits the labels included in the selected legend, along with their attributes
Edit Layout	Specifies the position of the labels in relation to an instrument

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Label Legends
Editing Label Legends
Formatting the Label Legend
Importing Label Legends from Another File
Repositioning Labels

Creating Label Legends

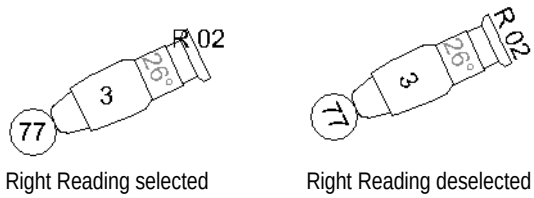
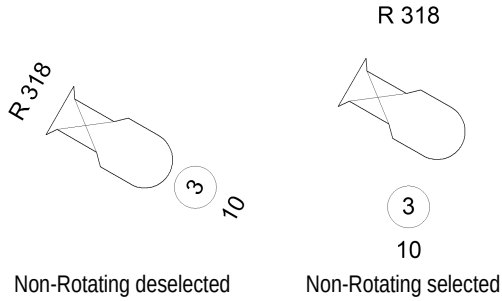
To create a new legend:

1. In the Label Legend Manager dialog box, click **Add**.

   The Add New Legend dialog box opens. Select the labels for placement and specify their attributes. Indicate the instrument symbol to use when formatting the label legend, and select whether the legend rotates with the instrument.

Click to show/hide the parameters.

Parameter	Description
Legend Name	Provide a name for the new legend by typing it in the Legend Name field. A name must be entered before closing the dialog box.
Attributes	
Use	Click to select an item to appear in the label legend. A check mark indicates the label is included. Click again to remove the label from use.
Attribute	Lists the possible labels that can be selected for inclusion in the label legend
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Right_Reading	For each label, select whether it will always be right-reading or allowed to rotate with the instrument. A check mark indicates the field is right-reading; deselect to allow the field to rotate with the instrument.  Right Reading selected Right Reading deselected
Container Type	Select the container for each label by clicking in the **Container** field until the desired container is displayed.
Lighting Instrument Layout Symbol	Specifies a symbol to use for label placement when editing the label layout; by default, the active symbol is selected. Click **Choose** to open the Choose an Instrument Symbol dialog box, where a different symbol can be selected from the current file's resources.
Non-Rotating	Specifies whether the label legend rotates with the instrument, or remains stationary as the instrument rotates  Non-Rotating deselected Non-Rotating selected

Custom container symbols can be added to the list of available containers. Draw a container object, and then click **Modify > Create Symbol**. Enter a name for the container; click **OK**. In the Move Symbol dialog box, specify the location of the Containers folder and click **OK**. The symbol is added to the list of available containers. Any symbol can be placed in the containers folder and used as a label container. The pre-defined container symbols can also be edited and customized.

Container attributes can be determined by the label text or lighting device; see "Lighting Device Setup" on page 95.

2. When all the label items and attributes have been specified for the legend, click **OK**. The new legend name displays in the **Legend Name** list in the Label Legend Manager dialog box.
3. Once the legend has been created, it needs to be formatted. See "Formatting the Label Legend" on page 871.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using the Label Legend Manager
Editing Label Legends
Formatting the Label Legend

Editing Label Legends

To edit a legend:

1. In the Label Legend Manager dialog box, select the legend, and then click **Edit Fields**. The Edit Label Legend dialog box opens. See "Creating Label Legends" on page 869 for a description of the legend parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To identify which legend needs to be edited before opening the Label Legend Manager, Right-click (Windows) or Ctrl-click (Mac) on the lighting device and select **Locate Label Legend in Resource Browser** from the context menu.

The legend name displays at the top of the dialog box, and cannot be changed from here.

To rename a label legend, locate it in the Resource Browser in the Label Legends folder of the current file (navigate there manually or use the **Locate Label Legend in Resource Browser** context menu command). When the label legend is highlighted, Right click (Windows) or Ctrl-click (Mac) and select **Resources > Rename**. (Any existing instruments that use the old legend name will lose their labels.)

2. The currently selected label items have a check mark next to the label name. Select any additional label items to appear in the legend by clicking in the **Use** column. To deselect a label item, click the associated check mark in the **Use** column; the check mark is removed.
3. Lighting Instrument Layout Symbol displays the name of the symbol currently used in the label legend. To use a different symbol, click **Choose** to open the Choose an Instrument Symbol dialog box. Select a new reference symbol from the resources present in the drawing and click **OK**.

   When you change the instrument's reference symbol, the symbol's appearance on the drawing will not change. However, the change will be reflected within the label legend; the name shown on the Edit Label Legend dialog box will update, as will the appearance of the symbol in the layout editor (see "Formatting the Label Legend" on page 871).
4. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using the Label Legend Manager
Creating Label Legends
Formatting the Label Legend

Formatting the Label Legend

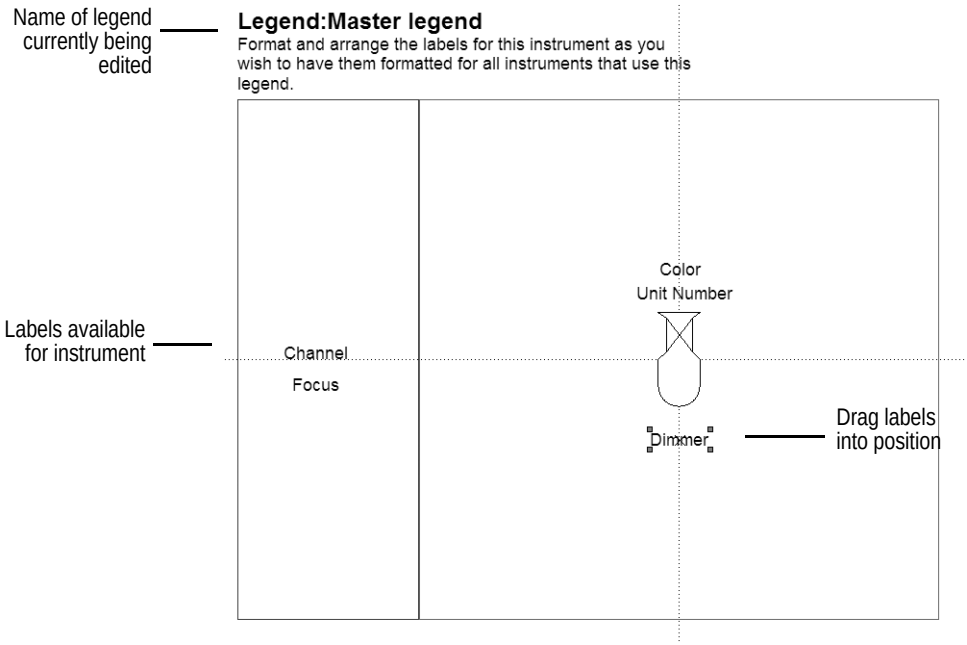
Once a label legend has been created, format it so that the labels are displayed in specific positions. The labels, containers, and layout symbol are selected when creating the label legend; see "Creating Label Legends" on page 869.

To format the label legend:

1. In the Label Legend Manager dialog box, select the legend, and then click **Edit Layout**.

   Alternatively, Right-click (Windows) or Ctrl-click (Mac) on a lighting device and select **Locate Label Legend in Resource Browser** from the context menu. The label legend associated with that lighting device is highlighted in the Resource Browser. Right-click (Windows) or Ctrl-click (Mac) on the highlighted label legend, and select **Edit** from the context menu. The Edit Symbol dialog box opens. Select **2D Component** and click **Edit**. (See "Editing Symbol Definitions" on page 247 for additional information about the Edit Symbol dialog box.)

   The Edit Symbol window opens, where the default position and format of the labels is specified for instruments that use this label legend. The name of the current label legend is displayed at the top of the Edit Symbol window. The instrument used for the layout is either the symbol selected in **Lighting Instrument Layout Symbol** or the active symbol at the time the legend was created.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



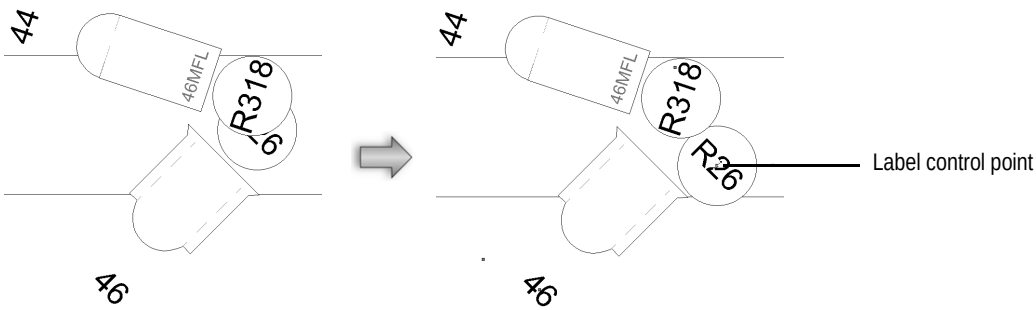
2. The labels selected for the legend display to the left of the instrument diagram. Drag each label to its position on or around the instrument. The label's formatting can be set by selecting the label, and then using the Object Info palette or the **Text** menu to select the label's font, size, style, and justification. The label stacking order can be changed with the **Modify > Send** commands; the instrument is part of the stacking order. Change the color of the label by selecting it and applying a solid pen color from the Attributes palette.

   If necessary, the labels placed outside the bounding box of the instrument are adjusted so they are always placed at an absolute distance from the instrument. For example, a label placed six inches in front of an instrument always appears six inches in front of any instrument to which the label is applied. The position of a label placed upon an instrument is scaled so it is in the same relative position whether the instrument is larger or smaller than the one used for formatting. A label placed at the center of the instrument is always at the center.

3. When the labels are set, click **Exit Symbol** in the upper right corner of the window to return to the drawing. The formatting and position of the labels applies to all instruments that use this legend.

Repositioning Labels

Labels should normally be repositioned through the Label Legend Manager. On a complex light plot, however, a label can be hidden by the nearest object. If that occurs, click on the instrument to select it, and then click on the instrument label control point, located at the center of the label, with the **Selection** tool. Drag the label to another location. This one-time repositioning does not affect the label legend.



Creating a Label Legend from an Existing Instrument

If labels have been repositioned and the arrangement is suitable to use as a label legend for other instruments, you can create a new label legend from a selected lighting instrument's legend. The current label arrangement and instrument rotation are used for the new label legend.

To create a label legend based on an existing instrument:

1. Select the lighting instrument. The instrument must already have a label legend attached.
2. Select **Spotlight > Label Legend > Create Label Legend from Instrument**.
3. Provide a unique name for the new label legend.

   By default, the instrument's existing label legend is appended with a -1 (or with the next available number) as a suggested name.
4. Click **OK**.

   The new label legend is created, and it is automatically assigned to the selected instrument. The instrument specified in **Symbol Name** in the Object Info palette of the selected instrument becomes the representative lighting device for the label legend.

Importing Label Legends from Another File

Label legends from another file created with the Vectorworks Spotlight product can be imported into the current file. Import the Label Legends symbol folder contents from the original file into the new file with the **Import** command in the Resource Browser; place the legends into the Legends folder. The imported label legends are listed in the Label Legend Manager. See "Importing a Symbol or Symbol Folder" on page 252.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Label Legends
Editing Label Legends
Using the Label Legend Manager

Adding Lighting Instruments

Instruments can be inserted from the thousands of symbols provided in symbol libraries, or created as custom symbols that you convert into lighting instruments.

- In the Resource Browser, either select one of the lighting symbols from the symbol libraries or one of the symbols you have created in this file or a Favorites file. Insert the symbol with the **Lighting Instrument** tool to place a lighting instrument object on the light plot. This method is described in "Inserting Instruments" on page 875.
- Convert custom geometry to a lighting instrument. This method is described in "Creating a Lighting Instrument Symbol" on page 874.

  The instrument symbols from the libraries included with the Vectorworks Spotlight product contain all the required attribute information. When creating or using your own symbols with the Vectorworks Spotlight product, see "Lighting Instrument Specifications" on page 890.

  An existing lighting instrument with specific parameters can be saved as a red symbol (a preformatted resource).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Lighting Instrument Symbol
Inserting Instruments

Creating a Lighting Instrument Symbol

Vectorworks Spotlight provides hundreds of lighting symbols; however, it is not necessary to use only these pre-defined symbols. Create a Spotlight instrument out of a currently selected symbol—once the symbol is converted to an instrument, the instrument functionality is present and the label legend is attached.

See “Lighting Instrument Specifications” on page 890 for more information and restrictions on using custom symbols for lighting instruments, and information on attaching the Light Info Record.

To create a lighting instrument:

1. Convert the object to a symbol by selecting **Modify > Create Symbol**.

   Symbols can have a separate 2D and 3D representation.The lighting instrument object automatically uses the appropriate portion of the symbol. At a minimum, the symbol must have at least a 2D component; it also requires a 3D component to display correctly in 3D views. For information on symbols, see “Creating New Symbols” on page 239.

2. Select the symbol or symbols.
3. Select **Spotlight > Object Conversion > Convert to Instrument**.

   The selected symbol is converted into a lighting instrument that remains inserted in the drawing. The active label legend, if any, is applied to the instrument(s); see “Using the Label Legend Manager” on page 868 for more information. The instrument properties can be changed through the Object Info palette; some of the instrument parameters may need to be entered.

   If creating multi-circuit instruments, first create each instrument’s symbol (see “Multi-circuit Instrument Specifications” on page 894). Then insert the multi-circuit instruments as described in “Instrument and Accessory Specifications” on page 890.

   Select the instrument from the Resource Browser as an active symbol definition and use the **Lighting Instrument** tool to insert again if required (see “Inserting Instruments” on page 875).

Creating a Preformatted Lighting Resource

Any instrument that has been inserted from a library or converted from a symbol can be saved as a red symbol. Its appropriate default values are retained, and when inserted from the Resource Browser, the instrument picks up the lighting position and active label legend. For more information on red symbols, see “Symbol Types” on page 237.

To create a lighting instrument resource with pre-defined parameters:

1. Select a current instrument and ensure that all its parameters have been specified as desired.

   In the Object Info palette, set the **Use Legend** parameter to None and delete the lighting position (**Position** parameter). Set the Z height of the instrument to 0.

   Set a label legend name to use a specific label legend, rather than the current one, upon insertion. If the label legend is not in the file at the time of symbol insertion, it is automatically imported.

2. With the instrument selected, select **Modify > Create Symbol**.

   The Create Symbol dialog box opens. Provide a name for the new symbol. For information on symbols, see “Creating New Symbols” on page 239.

3. Select **Convert to Plug-in Object**, and then click **OK**.
4. Click **OK** to create the symbol.

   If the Move dialog box opens, specify the location of the resource.

5. The lighting symbol is saved in the Resource Browser as a red symbol. When the symbol is inserted, it functions as a plug-in object and the saved parameters are retained.

Inserting Instruments
Editing Lighting Instruments
Spotlight Setup
Lighting Instrument Specifications

Inserting Instruments

Use **Lighting Instrument** tool to insert an existing instrument symbol. This can be a symbol that you have created, located in a Favorites file or the current file, or it can be one of the symbols imported from the libraries included with the Vectorworks Spotlight software ([Vectorworks]\Libraries); see “Resource Libraries” on page 219. When the Vectorworks Spotlight product is installed, lighting instrument symbols are also provided as default content. Default content is automatically imported into the file and displays in the Resource Browser.

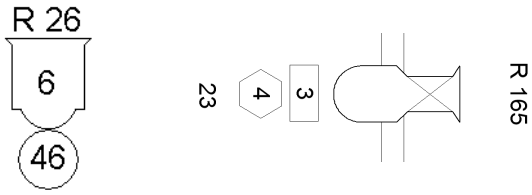
For information on importing symbols, see “Accessing Existing Resources” on page 229.

To insert a lighting instrument:

1. From the Resource Browser, double-click on the instrument symbol to insert. This automatically activates the symbol and selects the **Lighting Instrument** tool from the Spotlight tool set.

   Alternatively, select the instrument symbol from the Resource Browser and choose **Resources > Make Active** to make it the active symbol definition. The **Lighting Instrument** tool is automatically selected.

2. Click once to insert the active instrument symbol definition, and then again to determine the instrument rotation. By default, the active Label Legend is applied to the instrument.



   If an instrument symbol is accidentally inserted by the **Symbol Insertion** tool, the instrument functionality will not be present. When correctly inserted, the Object Info palette displays “Lighting Device” for the selected instrument.

   Once the **Lighting Instrument** tool has been selected, it remains selected by default so that instruments can be repeatedly inserted.

3. Place instruments on a lighting position to add them to that lighting position and allow auto-numbering. A different type of instrument can be placed by double-clicking on another instrument symbol definition and continuing to use the **Lighting Instrument** tool.

   Once instruments have been inserted, they continue to automatically associate with the nearest lighting position within a specified **Pick Radius** set in the Spotlight preferences. To avoid instruments changing their lighting position association as they are moved, disable automatic positioning.

Editing Lighting Instruments
Setting Up Instrument Label Legends
Inserting Lighting Positions
Spotlight Setup
Creating a Lighting Instrument Symbol

Inserting Multi-circuit Instruments

Lighting instruments can be inserted as multi-circuit instruments that act as a group while retaining their individual parameters and labels.

As an example, a three-cell Cyc light consisting of different colors can be inserted as a single multi-circuit instrument that can be easily positioned. However, each light in the multi-circuit instrument retains its own information, which is displayed in its label legend. When generating paperwork, each cell of the instrument is listed individually.

The **Lighting Instrument** tool inserts multi-circuit instruments.

The **Spotlight > Object Conversion > Convert to MultiCircuit** command is no longer required when inserting multi-circuit instruments, although it is still available.

To insert a multi-circuit instrument:

1. Create a multi-circuit symbol out of instrument symbols (left, center, and right components), or locate a multi-circuit symbol in the Resource Browser.

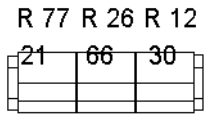
   See “Creating New Symbols” on page 239.

   Multi-circuit symbol

   The multi-circuit symbol must consist only of other symbols to be properly inserted as a multi-circuit instrument. Otherwise, the **Lighting Instrument** tool treats it as a single circuit instrument.

2. Click the **Lighting Instrument** tool from the Spotlight tool set.
3. Select **Resources > Make Active** to make the multi-circuit symbol the active symbol definition. Alternatively, double-click on the symbol to activate it.
4. Click once to insert the active multi-circuit instrument symbol definition, and then again to determine the multi-circuit instrument rotation. By default, the active Label Legend is applied to each part of the multi-circuit instrument. The multi-circuit instrument is numbered according to the lighting position settings.

   The instruments retain their individual information but move as a single unit

   The **Spotlight > Object Conversion > Convert to MultiCircuit** command can add a selected instrument to the multi-circuit instrument. To undo a conversion to a multi-circuit instrument, select the multi-circuit instrument and then the **Convert to MultiCircuit** command.

Inserting Lighting Positions
Inserting Instruments

Editing Lighting Instruments

Once lighting instruments have been inserted (normally, on a lighting position), many of the parameters of one or more selected instruments can be changed from the Object Info palette and Lighting Device dialog box, and the color, numbering, alignment, and other options can be controlled.

> To select all lighting instruments focused on a particular focus point, right-click (Windows) or Ctrl-click (Mac) on the focus point and select **Select Focused Lighting Devices** from the context menu. The focus point and any other selected objects are deselected automatically to facilitate faster editing of the lighting instrument(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Instrument Properties

Instrument (or accessory) parameters can be viewed and edited in the Object Info palette.

The parameters that display on the Shape tab of the Object Info palette depend on the settings made in Spotlight preferences; see “Specifying Lighting Device Parameter Display” on page 98.

Double-click on a lighting device, or click **Edit** from the Object Info palette or the context menu of one or more selected devices. The Lighting Device dialog box opens, facilitating instrument parameter modifications; see “Changing Instrument Properties” on page 880. Advanced light properties, which include turning the associated light source on, are available by editing the light source embedded within the instrument; see “Advanced Light Properties” on page 881.

For a custom instrument symbol without an information record attached, enter the required instrument object parameters (see “Attaching the Light Info Record and Light Info Record M” on page 893). An entry is not required for every field.

Click to show/hide the parameters.

Parameter	Description
XYZ/IJK Location/ Rotation	Changes the instrument’s location and rotation; the instrument labels rotate with the instrument unless **Right-Reading** was selected in the Label Legend Manager (see “Formatting the Label Legend” on page 871). In a 3D view, if an instrument is associated with a focus point, the instrument, along with any accessories, rotates automatically so that it points to the focus point.
Edit	Opens the Lighting Device dialog box, for editing the parameters of one or more lighting devices
Device Type	Displays the type of object that is selected; normally, instruments are Light device types (Moving Light, Device, Practical, SFX, Power, and Other are also considered Light types). Accessories such as color frames, barn doors, and top hats are Static Accessory types; accessories that require a control channel, such as color scrollers, are Accessory types.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Instrument Type	Displays the specific type of lighting instrument
Wattage	Indicates the power consumed by the instrument
Purpose	Specifies the purpose for using
Position	Displays the name of the lighting position
Unit Number	Identifies the instrument location on the lighting position
Color	Specifies the gel color number according to manufacturer code, RGB values, color name from the file, or web (hex) codes; when the light associated with the instrument is turned on, **Color** specifies the actual light color. Use + between color numbers to indicate combinations of two or more colors. See "Lighting Instrument Color" on page 882 The color setting can affect the display of the lighting device color, depending on the Spotlight preferences. See "Lighting Device Setup" on page 95
Dimmer	Specifies the dimmer or DMX address number of the instrument
Channel	Specifies the channel number of the instrument
Address	Specifies the absolute address of the instrument. If **Automatically assign Universe** is selected in the Spotlight Preferences dialog box, this field is used to automatically populate the **Universe** and **U Address** fields.
Universe	Indicates the DMX group of the dimmer; this is set automatically if the Spotlight file is set up to assign the universe. See "Universe Assignment in Vectorworks Spotlight" on page 99
U Address	When the universe is automatically assigned, displays the address assigned within the universe
Circuit Number	Indicates the circuit number where instrument is plugged in
Circuit Name	Specifies the name of bundled circuit group
System	Specifies the letter describing the control system (Lightwright-compatible parameter)
User Field 1 – 6	Provides user-defined fields; use these extra fields to keep track of any desired data (see "Specifying Lighting Device Parameter Display" on page 98)
Num Channels	Identifies the number of control channels used by the instrument
Frame Size	Indicates the dimensions of the color cut
Field Angle	Sets the field angle of the instrument's light beam
Field Angle 2	Specifies the second field angle for an elliptical light source
Beam Angle	Sets the beam angle of the instrument's light beam
Beam Angle 2	Specifies the second beam angle for an elliptical light source
Weight	Specifies the instrument weight
Gobo 1	Indicates the gobo texture number for the first gobo Specifying the gobo texture by clicking **Edit** in the Object Info palette provides a graphical method of selecting textures from the default content
Gobo 1 Rotation	Sets the rotation angle of gobo texture 1
Gobo 2	Indicates the gobo texture number for the second gobo
Gobo 2 Rotation	Sets the rotation angle of gobo texture 2
Gobo Shift	Adjusts the position of the instrument gobos to the front or rear

Parameter	Description
Mark	Provides a user-defined label (Lightwright-compatible parameter)
Draw Beam	Select to draw an accurate wireframe representation of the light beam; the light beam is drawn based on the instrument parameters, and can be used to check whether the stage and focus areas have been adequately lit (see “Drawing Light Beam Representations” on page 899). Spotlight setup can globally specify the appearance and display of the elements in the light beam (angles and centerline).
Draw Beam as 3D Solid	Generates a light beam that appears solid when rendered
Use Vertical Beam (Focus Point required)	By default, draws the wireframe beam in a vertical orientation when possible, when either **Draw Beam** or **Draw Beam as 3D Solid** is selected. When deselected, the beam is usually drawn in a horizontal orientation.
Show Beam at	In a 2D view, select whether to draw the beam as it crosses the focus point, at the falloff distance as projected through the focus point, or in both areas
Replace with Active Symbol	Replaces the selected instrument with the current active symbol (see “Replacing Instruments” on page 884)
Refresh Labels	Refreshes instrument labels if label changes have been made
Falloff distance	Specifies how far the wireframe light beam is drawn beyond the focus point; the **Focus Instruments** command sets this value to the distance between the focus point object and 0
Lamp Rotation Angle	Indicates the rotation angle of the virtual elliptical light source, from 0 to 90°
Shutter parameters	Controls the angle and depth of top, left, right, and bottom shutter cuts. Each shutter is located at 90º intervals around the lighting instrument, and can be adjusted +/- 45º. The shutter depth ranges from 0 to 100%, with 100% cutting through the center of the light source.
Symbol Name	Displays the name of the symbol used to create the instrument or accessory
Use Legend	Select the label legend to apply from the list of label legends in the file (see “Using the Label Legend Manager” on page 868)
Automatically Flip Front & Back Legend Text	Select to mirror the label legend along the X axis of the lighting instrument
Automatically Flip Left & Right Legend Text	Select to mirror the label legend along the Y axis of the lighting instrument
Focus	Specifies the focus point for the lighting instrument. The focus point must be defined first (see “Focusing Instruments” on page 897).
Set 3D Orientation	Specifies the X/Y position of the instrument at the hanging point, for 3D views only

For instruments and accessories, do not provide a name on the Data tab. The instrument or accessory Unique ID Number (UID) is automatically entered and must be used as the instrument or accessory name.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Changing Instrument Properties
Editing Lighting Instruments
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Specifying Lighting Device Parameter Display
Spotlight Setup
Advanced Light Properties

Changing Instrument Properties

To edit the properties of a lighting instrument or accessory:

1. Select one or more lighting instruments.
2. From the Object Info palette or the context menu, select **Edit**, or simply double-click a single instrument selection.

   The Lighting Device dialog box opens. The lighting device parameters are described in “Lighting Instrument Properties” on page 877. Only the parameters which are different are described here.

Click to show/hide the parameters.

Parameter	Description
Position	Lists the lighting positions in the file; select one for the lighting instrument, or select the ellipsis (...) to create a new lighting position name If a new position is named, a lighting position of that name must then be created in the file
Color	Specifies the gel color number according to manufacturer code, RGB values, color name from the file, web (hex) codes, color combination, or the color selected from the color box (see “Applying Colors” on page 1132). The selected color is added to the file’s active document color palette, and can be specified for a selected lighting device by its name. When the light associated with the instrument is turned on, **Color** specifies the actual light color (see “Lighting Instrument Color” on page 882). The color setting can affect the display of the lighting device color, depending on the Spotlight preferences. See “Lighting Device Setup” on page 95.
Focus	Lists the focus points in the file; select one for the lighting instrument, or select the ellipsis (...) to create a new focus point name If a new focus point is named, a focus point of that name must then be created in the file
Use Legend	Select the label legend to apply from the list of label legends in the file (see “Using the Label Legend Manager” on page 868)
Symbol Name	Lists the instrument or accessory symbols from the current file’s resources; select a different symbol for the lighting instrument or accessory from the resources imported into the file
Get Resource	Opens the Import External Resource dialog box; select an instrument or accessory symbol from either the default content or the current file’s content
Previous/ Next	When more than one device is selected, displays the parameters of the previous or next device in the selection
Apply to All	When more than one device is selected, and the parameters of the first device in the selection are displayed, applies only the parameters that have been modified to all the devices in the selection

3. Click the Light Information tab.

Click to show/hide the parameters.

Parameter	Description
Get Data from Symbol	Obtains the light information from the Light Info record attached to the symbol definition
Get Resource	Opens the Import External Resource dialog box; select a gobo texture symbol from either the default content or the current file's content (see "Inserting Gobo Projections" on page 973)

4. Click the Shutters tab to set the shutter cut information.

   Specify the depth and angles for the top, left, right, and/or bottom shutters. Each shutter is located at 90º intervals around the lighting instrument, and can be adjusted +/- 45º. The shutter depth ranges from 0 to 100%, with 100% cutting through the center of the light source.

5. Click the User Data tab to edit user field value information.

   User data fields and any default values display here and the default values can be edited by selecting a field name and entering the new value in **Value**.

6. Click **OK** to apply the changes to the selection.

The properties of one or more selected devices or objects can also be edited from the Shape tab of the Object Info palette (see "Lighting Instrument Properties" on page 877).

A selected lighting device or a selection of devices can be cut, copied, and pasted with the **Edit** menu commands. See "Editing Objects" on page 997 for more information on the basic editing commands.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Instrument Properties
Advanced Light Properties
Drawing Light Beam Representations
Editing Lighting Instruments
Spotlight Setup
Resource Libraries

Advanced Light Properties

More advanced control over the lighting instrument properties is possible by editing the light source embedded within the instrument. Turn the lighting instrument on and off by editing the embedded light.

To edit the embedded light source:

1. Select the lighting instrument(s).
2. From the lighting instrument context menu, select **Edit Light**.

   The Light Properties dialog box opens. These parameters offer full control over the light source, and are described in "Light Source Properties" on page 1578. The Renderworks product is required for certain parameters.

   Various light parameters are disabled, because they are controlled by the lighting device object instead.

   Select a light **Kind** of Custom to associate a light distribution file with the light. Note that light distribution files and theatrical lighting manufacturers may not specify output in the same way.

3. To enable the light properties, turn the light on in the Properties dialog box or from the Visualization palette, or right-click (Windows) or Ctrl-click (Mac) on the instrument and select **Turn On** from the context menu.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Instrument Properties
Light Source Properties

Drawing Light Beam Representations
Editing Lighting Instruments

S Lighting Instrument Color

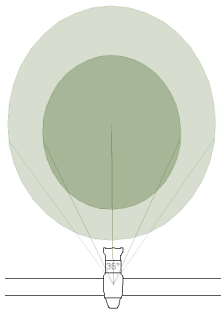
The gel color of a lighting instrument is specified by the **Color** parameter in the Object Info palette or Lighting Device dialog box. Define the color by one of these methods.

- RGB values separated by commas (such as 191, 49, 26); values must be between 0 and 255
- Manufacturer color codes, by manufacturer name or abbreviation and color number (such as Gam 650, G 650, G-650, or G650)
- Name of the color as specified in the file's color palettes (see "Applying Colors" on page 1132)
- Web hex values prefixed by the pound sign (such as #003366)
- Combinations of two or more colors (such as R46+R10); the resultant calculated color combination displays, just as when combining actual color filters. Invalid single colors are treated as white when calculating color combinations.

Acceptable manufacturer names and abbreviations are shown in the following table.

Manufacturer	Product Name	Abbreviation
Rosco	Roscolux	R, X
Rosco	Supergel	R, S, RS
Rosco	E-Color	E, RE
Rosco	CalColor	R, RC
Rosco	Cinegel	R, RCG
Rosco	Cinelux	R, RCL
Rosco	Storaro	R, RSO
Lee Filter	Lee	L
Gam	Gam	G
Apollo	Apollo	A, AP
GoboMan	GoboMan	GM

To draw a colored light beam for the lighting instrument, see "Drawing Light Beam Representations" on page 899.



The color setting can affect the display of the lighting device color, depending on the Spotlight preferences. See "Lighting Device Setup" on page 95.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Instrument Properties
Editing Lighting Instruments
Drawing Light Beam Representations
Spotlight Setup

Instrument Numbering

Instruments can be numbered automatically according to lighting position order, or manually according to specified settings.

Numbering Instruments Automatically

Instruments can numbered according to the lighting position automatic numbering parameters).

To automatically number the instruments on a lighting position:

1. Select the lighting position.
2. From the Object Info palette, specify the **Starting Number**, **Increment**, **Numbering Direction**, and, for multi-circuit instruments, the **Multi-Circuit Numbering**.
3. Click **Auto Number**.

   The instruments are automatically numbered. The **Unit Number** in the Object Info palette of a selected lighting instrument displays its number, which can be changed to override the automatic number.

   To display the numbers on the drawing, specify a label legend that includes the **Unit Number** in the label. See “Setting Up Instrument Label Legends” on page 868.

To automatically number all instruments assigned to a lighting position:

Select **Spotlight > Auto Number Positions** to globally automatically number the lighting instruments in the drawing based on the settings in each lighting position.

If lighting positions are selected when the command is activated, only the instruments on the selected lighting positions are numbered.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Numbering Instruments Manually
Inserting Lighting Positions
Editing Lighting Instruments

Numbering Instruments Manually

Instruments do not have to be numbered according to the lighting position’s automatic numbering feature. The **Number Instruments** command provides a method of numbering instruments by unit number and other parameters that automatically increment. In addition, the command can automatically increment several other instrument parameter fields.

Numbering Selected Instruments

To number selected instruments manually:

1. Select the instrument(s) to be numbered.
2. Select **Spotlight > Number Instruments**.

   The Number Instruments dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Field Name	Select the field name to automatically increment Universe is not available when automatic universe assignment is enabled; see “Universe Assignment in Vectorworks Spotlight” on page 99.
Prefix/Start#/ Suffix/ Increment	Specify a prefix for the numbering if desired, the starting number, an optional suffix, and the increment value. For example, a **Starting Number** of 2, **Increment** of 2, and **Suffix** of A numbers the instruments as follows: 2A, 4A, 6A, and so on.
Use # Chans	Instead of specifying a constant increment value, the increment can be based on the number of channels assigned to the instrument
Direction	The numbering direction is relative to the page; click the desired numbering direction. Select Manual to number the instruments by clicking, instead of by selection; see “Numbering Instruments by Clicking” on page 884.
Multi-Circuit Numbering	Select the numbering format for multi-circuit instrument numbering; these are identical to the selections made when specifying automatic numbering for multi-circuit instruments (see “Inserting Lighting Positions” on page 866)

3. Click **OK**.

   The selected instruments are numbered, and any fields automatically specified, in the selected direction.

Numbering Instruments by Clicking

To number instruments manually in the order they are selected:

1. Select **Spotlight > Number Instruments**.
2. Specify the field and numbering parameters as described in “Numbering Selected Instruments” on page 883.
3. In the **Direction** list, select Manual (the manual direction is selected automatically if no instruments were selected).
4. Click **OK**, and click the instruments to be numbered one by one. To stop numbering instruments, click in an empty area of the drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Instrument Numbering
Editing Lighting Instruments

Replacing Instruments

An individual instrument can be replaced by another type of instrument by choosing the new active symbol in the Resource Browser, and then selecting **Replace with Active Symbol** in the Object Info palette.

However, this process would be time-consuming for multiple instruments. By using the **Replace Instrument** command, all the instruments of a particular type can easily be replaced with another type of instrument. Alternatively, a selection of instruments can be replaced with another type of instrument.

Replacing All Instruments of One Type

To replace all the instruments of one type:

1. Select **Spotlight > Replace Instrument**.

   The Replace Instruments dialog box opens.

2. Click **Replace All**, and then specify the instrument type to be replaced by selecting it from the list.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. In the **Symbol Folders** list, select the location of the replacement instrument type. Select the specific instrument type from the **Symbols** list.
4. Click **OK**. All instruments of the type specified are replaced with the selected instrument type.

Replacing Selected Instruments

To replace selected instruments:

1. Select the instruments to be replaced.
2. Select **Spotlight > Replace Instrument**.

   The Replace Instruments dialog box opens.
3. Click **Replace selected instruments**.
4. Specify the location of the replacement instrument type in the **Symbol Folders** list. Select the specific instrument type from the **Symbols** list.
5. Click **OK**. The selected instruments are replaced with the specified instrument type.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Lighting Instruments
Inserting Instruments

Changing the Instrument Label Legend

As instruments are placed on the light plot, the active label legend is assigned to them by default. However, the legend can be changed for a selection of instruments.

To change the instrument label legend:

1. Select one or more instruments.
2. Click on the **Spotlight > Label Legend > Assign Legend to Instruments** command.

   The **Assign Legend to Instruments** command is also available from the object context menu for selected instruments. Right-click (Windows) or Ctrl-click (Mac) to quickly access the command.

   The Assign Legend to Instruments dialog box opens.
3. Select the new label legend for the instruments.
4. Click **OK**; the new label legend is applied to the selection.

   The label legend can also be changed for an individual instrument by selecting the instrument, and then selecting the legend name from the **Use Legend** list in the Object Info palette.

   Use the **Find and Modify** command to quickly generate a custom selection of specific instruments and apply the new legend to the selected instruments (see “Find and Modify” on page 887).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using the Label Legend Manager
Adding Lighting Instruments

Refreshing Instrument and Light Position Data

The **Spotlight > Refresh Instruments** command updates light position information and instrument labels to reflect any changes that have been made. This ensures that all current data is displayed; it is a good idea to select this command before printing the light plot.

Two-fers do not get updated with the instrument information when the **Refresh Instruments** command is selected.

The **Refresh Instruments** command is also available from the document or object context menu. Right-click (Windows) or Ctrl-click (Mac) to quickly access the command.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Lighting Instruments
Using the Label Legend Manager
Adding Lighting Instruments

Aligning Instruments

Lighting instruments can be automatically aligned and distributed along a specified guide line with the **Align and Distribute Items** tool.

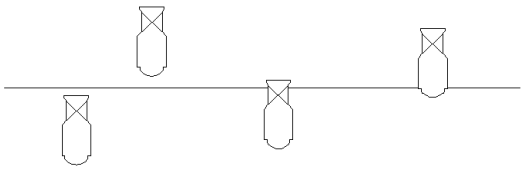
To align and/or distribute instruments along a line:

1. Select the objects to be aligned or distributed.
2. Click the **Align and Distribute Items** tool from the Spotlight tool set.
3. Click to define the beginning of the guide line, and click to end.

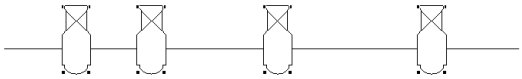
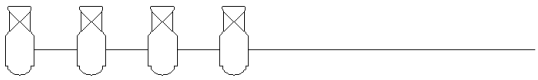
   The Align and Distribute dialog box opens.

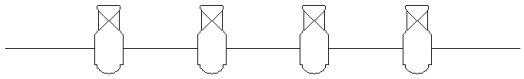
   The instruments can be aligned and distributed according to several options.

Original instrument alignment and distribution



Click to show/hide the parameters.

Parameter	Description
Just Align	The instruments are aligned, but their distribution is not changed 
Along line on centers	Aligns and distributes selected objects along the guide line using a specified spacing setting between the center point of the instruments 
Evenly between points	The instruments are aligned and distributed evenly between the guide line points, starting with instruments at the beginning of the guide line 
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Evenly inside points	The instruments are aligned and distributed evenly between the guide line points, starting with spacing at the beginning of the guide line 

4. Click **OK** to align or distribute the instruments according to the selected option.

Editing Lighting Instruments
Adding Lighting Instruments

Find and Modify

Custom selection criteria can be defined to quickly and easily search for instruments, lighting positions, and accessories and to perform an action upon the search results.

To find and modify objects:

1. Select **Spotlight > Find and Modify**.

   The Find and Modify dialog box opens.

2. Select the type of object to be searched and specify the change(s) to occur to the items found.

Click to show/hide the parameters.

Parameter	Description
Find All	Select the type of object to be searched, and then specify the criteria for the search. For example, find all instruments with a circuit number of 66. The list of values is filtered as the criteria are defined.
Change the __ field to __	A specific parameter value can be changed to a new value by clicking **Change**, selecting the parameter, and entering the new value. Instruments on a particular circuit can be changed to another circuit, for example.
Replace them with this symbol	Click **Replace** to change the found items to the item specified in the list. For example, find all instruments with a particular purpose, and change them to be of the same instrument type.
Assign this Label Legend	The label legend for found instruments can be changed by clicking **Assign** and selecting the new label legend
Renumber the __ field from __ starting at__	The renumber operation changes the numbering values for the specified item and location. Enter the starting number value for the change. For example, change the numbering for all PAR instruments on Electric #2 to begin with 12.
Copy the __ field to the __ field	One parameter from the searched item can be copied to another parameter by selecting the parameters from the lists in the **Copy** operation. For example, copy the circuit number to the dimmer number or the dimmer number to the channel number.
Change the class to __	Change the class of the found items by clicking **Change the Class to**

Parameter	Description
Just Select Them	The found item(s) can simply be selected by clicking **Just Select Them**
Just Deselect Them	Deselect the items found by the custom search by clicking the **Just Deselect Them** option
Delete Them	Delete the items found by the custom search by clicking the **Delete Them** option
Item Count	Displays an item count of the objects meeting current criteria at the bottom of the Find and Modify dialog box

3. Once the find and modify criteria have been defined and action(s) specified, click **OK**. The selected change(s) will occur to the items located by the search.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Lighting Positions
Adding Lighting Instruments
Adding Accessories

Adding Accessories

Accessories can be inserted from the many symbols provided in symbol libraries, or created as custom symbols that you convert into accessories.

- In the Resource Browser, either select one of the accessory symbols from the existing symbol libraries or one of the symbols you have created in this file or a Favorites file. Insert the symbol with the **Lighting Accessory** tool to place the accessory object on the light plot. This method is described in “Inserting Accessories” on page 889.
- Convert custom geometry into an accessory. This method is described in “Creating an Accessory” on page 888.

  The accessory symbols from the libraries included with the Vectorworks Spotlight product include all the required attribute information and have the correct accessory type assigned (Accessory or Static Accessory). To create or use your own symbols, see “Accessory Specifications” on page 894.

Lighting instrument accessories, such as color frames, barn doors, and top hats, are placed on the same design layer as the instruments. Accessories such as color frames, barn doors, and top hats have a **Device Type** of Static Accessory. Accessories that require a control channel, such as color scrollers, have a **Device Type** of Accessory.

An instrument can have multiple accessories in the same location (most accessories are placed at the front of an instrument). It can also have several accessories in different locations. Once an accessory has been associated with an instrument, the instrument controls the accessory.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating an Accessory
Inserting Accessories

Creating an Accessory

The Vectorworks Spotlight program provides accessory symbols; however, it is not necessary to use only the pre-defined symbols. Create an accessory out of a currently selected symbol—once the symbol is converted to an accessory, the accessory functionality is present. See “Accessory Specifications” on page 894 for restrictions on using symbols for accessories.

To create an accessory:

1. Draw the accessory, and then convert the object to a symbol by selecting **Modify > Create Symbol**.

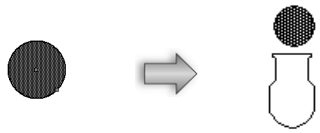
Symbols can have a separate 2D and 3D representation. The accessory object automatically uses the appropriate portion of the symbol. At a minimum, the symbol must have at least a 2D component; it also requires a 3D component to display correctly in 3D views. For information on symbols, see “Creating New Symbols” on page 239.

2. Select the symbol(s).
3. Select the lighting instrument to associate with the accessory or accessories.

   Only one instrument can be selected.
4. Select **Spotlight > Object Conversion > Convert to Accessory**.

   The selected symbol is converted into an accessory and remains inserted in the drawing. It is associated with the selected lighting instrument. The accessory properties can be changed through the Object Info palette; some of the accessory information may need to be entered.

   Accessories such as color frames, barn doors, and top hats should have a **Device Type** of Static Accessory. Accessories that require a control channel, such as color scrollers, should have a **Device Type** of Accessory.



   Select the accessory from the Resource Browser as an active symbol definition and use the **Lighting Accessory** tool to insert it again if required (see “Inserting Accessories” on page 889).

   To associate an accessory with a different lighting instrument, or associate an accessory that was never properly associated, select the accessory and the instrument to associate, and then select **Spotlight > Object Conversion > Convert to Accessory**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Accessories
Adding Accessories

Inserting Accessories

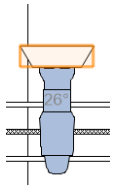
Existing accessory symbols are inserted with the **Lighting Accessory** tool. This can be a symbol that you have created, located in a Favorites file or the current file, or it can be one of the symbols imported from the libraries included with the Vectorworks Spotlight software ([Vectorworks]\Libraries); see “Resource Libraries” on page 219.

For more information on importing symbols, see “Importing a Symbol or Symbol Folder” on page 252.

 To insert an accessory:

1. From the Resource Browser, double-click on the accessory symbol to insert. This automatically activates the symbol and selects the **Lighting Accessory** tool from the Spotlight tool set.

   Alternatively, select the accessory symbol from the Resource Browser and choose **Resources > Make Active** to make it the active symbol definition. The **Lighting Accessory** tool is automatically selected.
2. Click once to insert the active accessory symbol definition, and then again to determine the accessory rotation. The cursor changes to a bull’s eye. Click on the instrument which will be associated with the accessory.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Because the accessory is controlled by the associated instrument, the items move together. However, an accessory can be positioned on its own by selecting it and dragging.

If an accessory symbol is accidentally inserted by the **Symbol Insertion** tool, the accessory functionality will not be present. If correctly inserted, the Object Info palette displays "Lighting Device" (with a **Device Type** of Accessory or Static Accessory) for the selected accessory.

Once the **Lighting Accessory** tool has been selected, it remains selected by default so that accessories can be repeatedly inserted. To place non-instrument symbols after placing accessories, switch to the **Symbol Insertion** tool.

Accessory Properties

The accessory Object Info palette parameters are nearly identical to those of an instrument, because it requires many of the same parameters. The **Device Type** is Static Accessory or Accessory.

Entering a parameter in the accessory Object Info palette will not change the associated instrument parameters.

For more information on the Object Info palette parameters, see "Lighting Instrument Properties" on page 877.

When a lighting instrument is deleted, its associated accessory is also deleted.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Accessories

Instrument and Accessory Specifications

Custom symbols can be created and used in addition to the pre-defined standard symbol sets. Create an instrument, accessory, or lighting position out of a currently selected symbol by converting it with the commands in the **Spotlight** menu.

Certain rules apply when creating lighting and accessory symbols.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Instrument Specifications
Multi-circuit Instrument Specifications
Accessory Specifications

Lighting Instrument Specifications

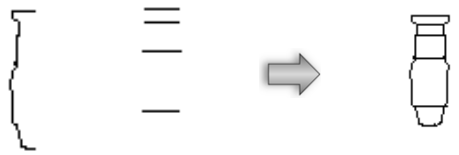
Special rules apply when creating a symbol to be converted to a lighting instrument.

Symbol Characteristics

Symbols should be hybrid (2D/3D) so that they display properly in both 2D and 3D views. At a minimum, the symbol must contain a 2D component, which must be a screen plane representation and not a 2D planar object.

2D Characteristics

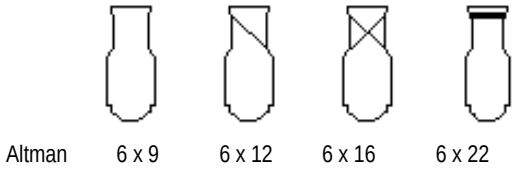
Create the 2D view of the symbol using as few polygons and lines as possible. If possible, use a single polyline rather than individual line segments. All instruments should be drawn with the front of the instrument (the end which emits light) oriented towards the top of the drawing. The symbol below was created from these few constituent parts:



The line weight of the symbol is also a consideration; the instruments need to stand out when printed. The outer perimeter of the symbol should have a line weight of at least 1/2 point (7 mils). Interior details should use a lighter line weight.

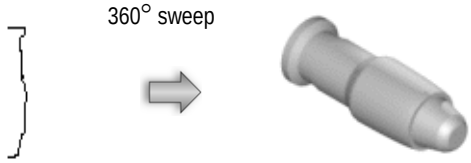
The 2D representation should have a solid fill so that it obscures information under the symbol. The size of the instrument should be accurate based on the real instrument it represents. While drawing the instrument, keep the level of detail as minimal as possible. The goal is to be able to distinguish instruments from one another, not to create a detailed plan view of each instrument.

For instruments with multiple configurations, it is acceptable to use simple graphical differences to distinguish among the models. For example, use the following variations to separate the different versions of a symbol:



3D Characteristics

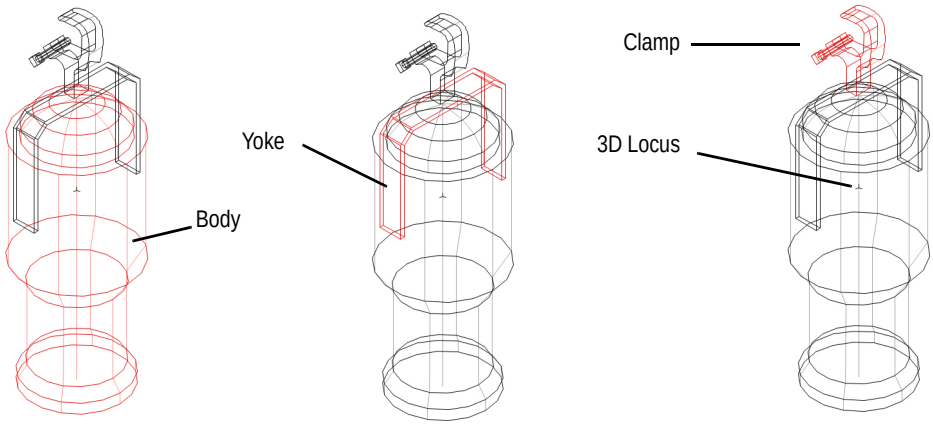
The 3D portion of the instrument should be drawn as if it is hung straight down (along the Z-axis) with the yoke oriented along the Y-axis. The top of the instrument should be oriented towards the top of the drawing. An easy way to generate a reasonable 3D instrument body is to sweep the 2D portion of the symbol. The segment angle of the sweep should be between 20 and 40°. See “Sweeping Objects” on page 1042 for more information.



Keep the 3D symbol simple. It should be solid. The model should be accurately sized, but without minute details like handles, grommets, fins, louvers, cords, and knobs. These items can add significantly to the rendering time required, and are not necessary to distinguish among instruments.

The 3D model should consist of three parts: the body, a yoke, and a clamp or base. Any subparts should be made into a single object or group for each of these pieces. The body represents the part of the instrument where light is emitted; the yoke connects the body to the base, and the base consists of either a base motor unit for moving lights or a clamp or other hanging device for other lights. The clamp can be imported from the symbol library provided with the Spotlight program. Place a 3D locus within the body of the instrument.

All the parts should be aligned as shown; the yoke rotates about the Z-axis, and the body rotates at the height of the locus point.



Insertion Point

Align the 2D and 3D views so that the hanging points of both versions line up.

The insertion point of the 2D/3D hybrid symbol in Top/Plan view should represent the hanging location of the instrument. The 3D insertion point should be the hanging point (center of the clamp or base).

Create the symbol as described in “Creating New Symbols” on page 239.

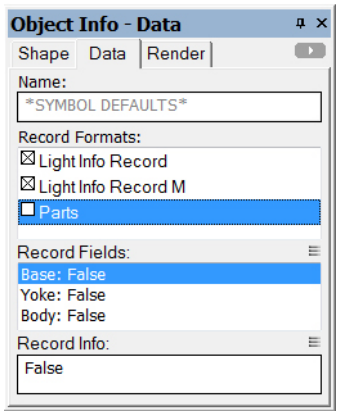
Attaching the Parts Record

Attach the Parts record to each of the three pieces of the 3D instrument model: body, yoke, and base or clamp.

To attach the Parts record:

1. In the Resource Browser, import the Parts record from one of the instrument library files included with the Spotlight program.

   Locate the record and select **Resources > Import** to bring the record into the current file.
2. Select the new symbol, and select **Modify > Edit Symbol**.
3. In the Edit Symbol window, select the body, yoke, or base/clamp.
4. Click on the Data tab in the Object Info palette. Attach the Parts record to the part by selecting the check box.



5. Edit the record by selecting the appropriate record field and changing its value to True. For example, select the yoke, attach the Parts record, and change the Yoke record field to True.
6. Repeat steps 3 – 5 for each of the three parts of the instrument model.
7. Click **Exit Symbol** at the upper right corner of the window to return to the drawing.

Attaching the Light Info Record and Light Info Record M

Attach the Light Info record to the lighting instrument symbol, with field names that match the names of the fields in the instrument object. Not all the fields are required, but the desired fields for the instrument object to read should be included. Filling the instrument type field with the manufacturer's name and model name for the particular instrument is recommended.

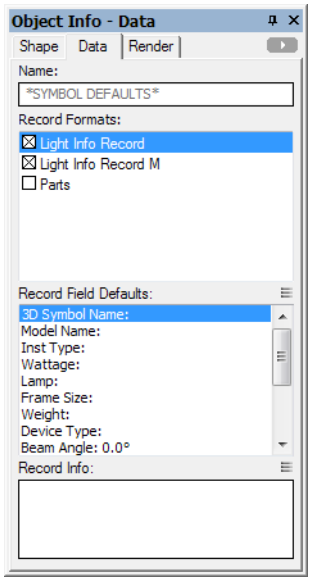
The Light Info M record provides metric measurements of the weight and frame size of the instrument object. This is needed for instruments that could be used in either imperial and metric drawings.

To attach the light info record:

1. In the Resource Browser, import the Light Info Record and Light Info Record M from one of the instrument library files included with the Spotlight program.

   Locate the records and select **Resources > Import** to bring the records into the current file.

2. Select the new symbol, and select **Modify > Edit Symbol**.
3. In the Edit Symbol window, click on an empty location so that nothing is selected.
4. Click on the Data tab in the Object Info palette. Attach the Light Info Record and Light Info Record M to the symbol defaults by selecting the check boxes. Then edit the records by selecting the record field and entering its record information.



   The **Candlepower**, **Beam Angle**, and **Field Angle** parameters affect the photometric grid and photometer object calculations. The **Beam Angle** and **Field Angle** parameters affect the Draw Beam feature.

   Normally, do not include text labels with the instrument, as these are handled by the instrument object. An exception can be made to distinguish different models or lamps of an instrument. For example, create three versions of a single PAR64 symbol by adding MFL, WFL, and NSP text blocks.

   Symbols should be named with the model name of the lighting instrument.

5. Turning the lighting instrument's light on in the Visualization palette includes a spot light as part of the lighting instrument. While editing the symbol, the spot light can be added, and accurate lighting information specified with the parameters in **Use Emitter**. See "Adding Light Sources" on page 1574 for information on adding a spot light and setting accurate lighting parameters.
6. Click **Exit Symbol** at the upper right corner of the window to return to the drawing.

As an alternative to the process of manually editing the symbol definition to attach the Light Info (and/or Light Info M) record, use the **Lighting Symbol Maintenance** command as described in "Lighting Symbol Maintenance" on page 959. Add the symbol by clicking **New**, and the Light Info record is automatically attached to it. Once in the maintenance list, the record data can be easily edited.

Multi-circuit Instrument Specifications
Accessory Specifications

Multi-circuit Instrument Specifications

Multi-circuit instruments represent a special case of the instrument object. Each symbol drawn should represent a single element of the multi-circuit instrument. An instrument that is at the end of the multi-circuit strip should look different from the instruments that are in the center. Place the number of symbols required to create the multi-circuit instrument. For example, place four symbols (two ends and two middle) to create a four-circuit cyc unit.

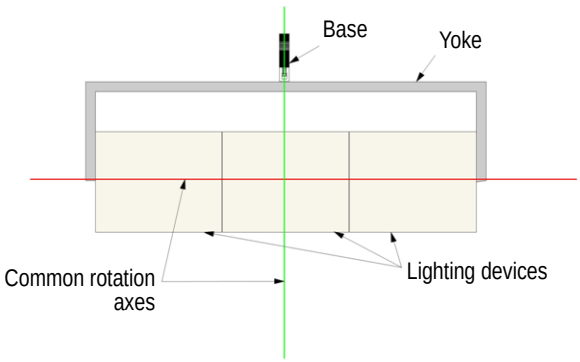
1. Convert symbols to instruments

2. Convert instruments to multi-circ instrument

The multi-circuit symbol must consist only of other symbols to be properly inserted as a multi-circuit instrument. Otherwise, the **Lighting Instrument** tool treats it as a single circuit instrument.

Attach the light info record to the individual instruments within the multi-circuit unit by the same method as for an individual lighting instrument.

The multi-circuit lighting device focuses in 3D in the same way as individual lighting devices (see "Lighting Instrument Specifications" on page 890). The yoke is created from portions of the individual instrument symbols. The 3D locus, placed within the container multi-circuit symbol, is used as a common rotation axis for the entire multi-circuit unit.



Lighting Instrument Specifications
Attaching the Light Info Record and Light Info Record M
Inserting Multi-circuit Instruments

Accessory Specifications

The accessory symbol should be a 2D/3D hybrid object. At a minimum, the symbol must contain a 2D screen representation. Keep the accessory representation as simple as possible to reduce rendering time.

The 3D portion of the accessory should be drawn as if it is hung straight down (along the Z-axis). The top of the accessory should be oriented towards the top of the drawing. 3D geometry should be drawn below the Z-axis to properly align with the front of the lighting instrument.

The Object Info palette for accessories and lighting instruments looks identical, but the **Device Type** differs for accessories (Accessory or Static Accessory).

Accessories should have a record attached for storing the default accessory values. The record should not contain fields that vary from instance to instance of an accessory. For example, do not add a Color Scroller channel field value unless that channel is used by all the color scrollers in the file.

The attached record should consist of the following fields:

Field	Description
Instrument type	Accessory name; this can be a specific manufacturer's model name or number, or a generic name for the accessory, such as 6" Top Hat
Accessory type	Generic accessory category; in some cases, this can be similar, or identical to the instrument type (example accessory types include Top Hat, Barn Door, Color Scroller, Gobo Rotator, and Color Frame)
Wattage	Amount of power consumed by the accessory
Weight	Accessory weight
Other	Add fields as desired; these fields are read into the Lighting Device object if the field name in the accessory record matches the field name of the Lighting Device object

A Parts record is not required for an accessory.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Instrument Specifications

Ganging Instruments

Two-fers gang two or more instruments together on one circuit, dimmer, channel, or dimmer and channel.

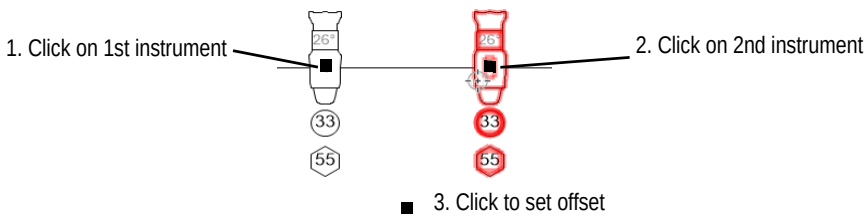
The ganging field parameter value(s) should match. If not, the value(s) from the first selected instrument are used.

To gang two or more instruments:

1. Click the **Ganging** tool from the Spotlight tool set.

   Click **Preferences** from the Tool bar to set the default tool preferences, if desired.

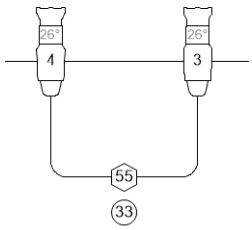
2. Click on the first, and then the second, instrument to be ganged together. If more than two instruments are to be ganged together, continue clicking on each instrument in succession.



3. Click to set the offset distance of the two-fer label.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

If the field values of the instruments do not match, confirm that this is acceptable in the Notice dialog box which opens.

4. Click **OK**. The instruments are ganged together with one or more two-fer objects.



The properties of a two-fer are displayed, and can be edited, in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Update Insts	Select to update the ganged instruments. Changes to one instrument are updated for the other ganged instrument. Also updates two-fer counts in the position summary inventory report.
Corner Style	Select the corner style for the two-fer object (Square, Arc, or Bézier)
Gang By	Indicate whether to gang by Channel, Dimmer, Circuit (name and number), or dimmer and channel
Field Value	Enter the value for the two-fer label. (For example, if ganging by channel number, enter a channel number of 80.) This new value updates the former value for dimmer or channel in the instrument record. If ganging by circuit or channel and dimmer, enter the two values separated by a comma.
Show Label	Select to display the two-fer label or labels with the ganged values
Use Container	Select to place the label or labels in a container; if ganged by dimmer and channel, the same container is used for both labels. By default, the label legend container from the first selected instrument is used, but this can be overridden by specifying a container name.
Container Name	The container name must exactly match one of the container symbol names located in the Resource Browser (normally, these are in the Containers folder)

Inserting Instruments

Focusing Instruments

To focus a lighting instrument assembly and beam on a particular area or object, a focus point needs to be defined.

The focus point can also be used to create Magic Sheets that show the instruments focused on a particular area (see “Magic Sheets” on page 970). In addition, the focus point can be used as one of the criteria for finding instruments with the **Find and Modify** command (see “Find and Modify” on page 887). It is also possible to select all lighting devices assigned to a particular focus point for quick editing (see “Editing Lighting Instruments” on page 877).

Creating a Focus Point Object

An object defined as a focus point specifies where instrument assemblies and light beams should be directed. If the focus point is moved, any instruments that are aimed at it update their focus position based on the focus point information.

Like lighting positions and instruments, focus points should be inserted on their own design layer, to facilitate selection, viewing, and printing. Alternatively, it is acceptable to insert the focus points on the same design layer as the scenic elements. Focus points can also be inserted into their own classes.

A 3D stage object can be used as a focus point for lighting instruments by naming it as the focus point in the instrument Object Info palette (see “Lighting Instrument Properties” on page 877).

To create a focus point object:

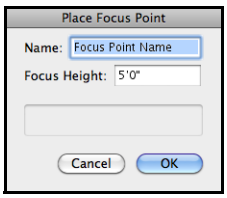
1. Click the **Focus Point** tool from the Spotlight tool set.
2. Click **Preferences** from the Tool bar to specify the default focus point settings. These settings can be changed later for existing focus points from the Object Info palette.

   The Focus Point Properties dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Show Focus Point Name	Select whether to display the focus point name along with the focus point shape
Focus Point Shape	Select a shape for the focus point. • **Standard Hybrid:** an octagon in both Top/Plan view and 3D views • **Standard 2D:** an octagon in Top/Plan view and a 3D locus in 3D views • **Standard 3D:** a 2D locus in Top/Plan view and an octagon in 3D views • **Locus Points Only:** a 2D locus point in Top/Plan view and a 3D locus in 3D views (does not display in rendered views) • **Default content:** Existing focus shape symbols are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219).

3. Click on the plot to insert the focus point object.
4. The Place Focus Point dialog box opens. Enter the name of the focus point, the focus height above the stage floor, and then click **OK**.



The focus point name is required later to specify the focus point for the lighting instruments.

The name of a focus point can be changed on the Data tab of the Object Info palette, and can be updated in the drawing file by selecting **Reset** on the Shape tab of the Object Info palette. Place focus points in their own class so they can be easily hidden for a 3D rendering. Alternatively, select Standard 2D or Locus Points Only for the **Focus Point Shape**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Assigning a Focus Point to a Lighting Instrument
Drawing Light Beam Representations

Assigning a Focus Point to a Lighting Instrument

One or more instruments and/or multi-circuit instruments can be focused on a focus point. The focus point is used to draw light beam representations (see "Drawing Light Beam Representations" on page 899), calculate photometric values (see "Obtaining Photometric Data" on page 899) and for rendering gobo projections (see "Gobo Projection Requirements" on page 976). In 3D views, the instrument rotates to point automatically at the focus point.

With the **Focus Instruments at Next Click** context menu command, selected lighting instruments can quickly focus on an existing focus point, or create one using the current focus point properties. Alternatively, use the **Focus Instruments** command to select an existing focus point, or create one and set its properties at that time.

To focus the lighting instrument(s) at the next click with the context menu command:

1. Select the instrument(s).
2. Right-click (Windows) or Ctrl-click (Mac) in an open area of the drawing, and select **Focus Instruments at Next Click** from the context menu.
3. To create a new focus point for the instrument(s), click in the drawing. The current focus point properties are used, and a unique name is automatically assigned to the new focus point; see "Creating a Focus Point Object" on page 897. Alternatively, click on an existing focus point or locus.

   The instrument(s) are assigned to the new or existing focus point.

   If you place a new focus point on an existing 3D object while in a 3D view, the focus point automatically adopts the Z value of the click.

To focus the lighting instrument(s) with the menu command:

1. Select the instrument(s).
2. Select **Spotlight > Focus Instruments**.

   Alternatively Right-click (Windows) or Ctrl-click (Mac) on the selected instrument, and select **Focus Instruments** from the context menu.

   The Focus Instruments dialog box opens. Select an existing focus point, or select **Next Click** to create a new focus point with the next mouse click and click **OK**.

   If **Next Click** is selected, the New Focus Point dialog box opens. Specify the name and height of the focus point.

   To change the focus point of a single instrument, enter the name of the new focus point in the **Focus** field of the Object Info palette (see "Lighting Instrument Properties" on page 877).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Focus Point Object
Drawing Light Beam Representations

Drawing Light Beam Representations

Once the focus point of instruments has been specified, light beam representations can be drawn. Wireframe light beam representations can be controlled by class; see “Lighting Device Setup” on page 95.

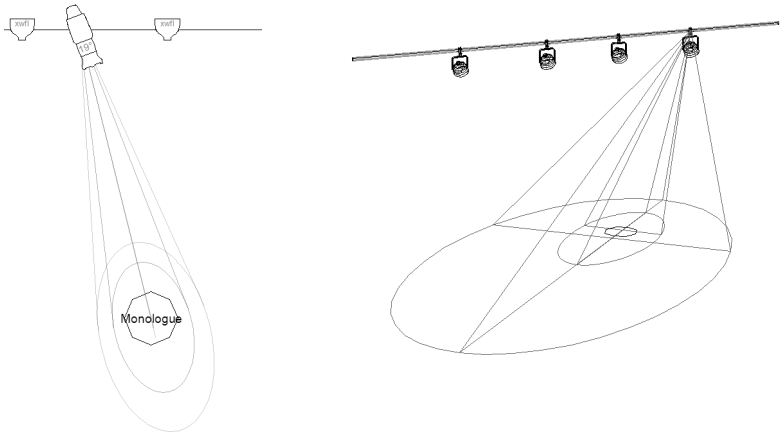
To turn on the light beam for one or more instruments:

1. Select the instrument(s).

   Use the **Select Focused Lighting Devices** context menu to quickly select all lighting devices assigned to a focus point (see “Changing Instrument Properties” on page 880).

   An instrument must have a focus point in order to draw a light beam representation (see “Assigning a Focus Point to a Lighting Instrument” on page 898). Elliptical light sources require secondary beam and field angles. The **Focus Instruments** command inserts a default falloff distance for elliptical light sources.

2. Select **Draw Beam** from the instrument Object Info palette (see “Lighting Instrument Properties” on page 877).



Light beam
Top/Plan view

Light beam
3D view

   An accurate wireframe representation of the light beam’s spread and location on the stage is drawn, oriented to the focus point. The light beam of instruments used for general wash lighting can also be drawn; however, a focus point is still required in order to draw the light beam representation.

3. Select **Draw Beam as 3D Solid** from the instrument Object Info palette to see the light beam as a solid cone of light. The **Color** specified in the Object Info palette or Lighting Device dialog box determines the solid color (see “Lighting Instrument Color” on page 882). The class of the light beam can also determine its appearance and optionally, texture (Renderworks required).

Creating a Focus Point Object
Assigning a Focus Point to a Lighting Instrument

Obtaining Photometric Data

Vectorworks can determine and display the surface illumination values of the stage at a specific location (Photometer object) or along a grid (Photometric Grid object).

For photometric values to be measured, the lighting instrument(s) must have a designated focus point.

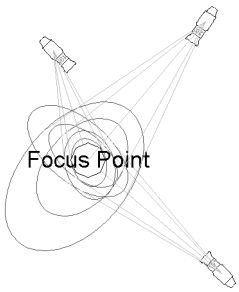
Inserting a Photometric Grid

The Photometric Grid is a is a three-click rectangular object, and can be inserted in Center-line Placement mode or Edge Placement mode.

Placement Mode	Description
Center-line	Click once, and then again, to define the width through the center of the grid. Click again to specify the width of half the grid.
Edge	Click once, and then again, to define the length along the edge of the grid. Click again to specify the grid width.

To insert a photometric grid:

1. Ensure that each lighting instrument contributing to the illumination is focused.



   The light beams do not have to be drawn for calculations to be made.
2. Click the **PhotoGrid** tool from the Spotlight tool set and click the desired placement mode from the Tool bar.
3. Click in the drawing area to insert the photometric grid.

   If this is the first time a photometric grid has been placed on the drawing, the object properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**.

   The photometric grid properties can be changed in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Show Elevation	Select to display the grid elevation along with the photometric values
Show Zero	Select to display elevation values of zero
Calculate Using Only Visible Beams	Select to determine the photometric values based on only instruments with **Draw Beam** selected; this reduces calculation time
Use Threshold Settings	Select to display the grid with colors that correspond to the specified threshold ranges
Lower Threshold	Specify the illumination value at the lower end of the range; values below this level are displayed in green, and values between the lower and upper threshold are displayed in tan
Upper Threshold	Specify the illumination value at the upper end of the range; values above this level are displayed in blue
Grid Spacing X	Specifies the spacing of photometric values in the X direction
Grid Spacing Y	Specifies the spacing of photometric values in the Y direction
Grid Width X	Specifies the length of the grid in the X direction; available in Object Info palette only
Grid Width Y	Specifies the length of the grid in the Y direction
Refresh	Refreshes the grid display after changes have been made to the lighting conditions

After placing the photometric grid, set the grid's Z value on the Object Info palette; illumination values vary depending on the grid's elevation.

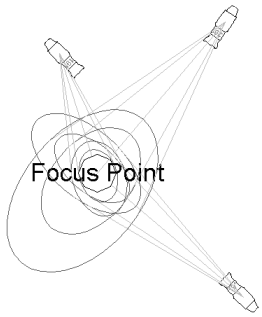
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Obtaining Photometric Data
Inserting a Photometer
Using Photometric Threshold Settings
Assigning a Focus Point to a Lighting Instrument

Inserting a Photometer

To insert a photometer:

1. Ensure that each lighting instrument contributing to the illumination is focused.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



The light beams do not have to be drawn for calculations to be made.

2. Click the **Photometer** tool from the Spotlight tool set.
3. Click to place the photometer in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
4. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Show Elevation	Select to display the object's elevation along with the photometric value
Show Zero	Select to display an elevation value of zero
Calculate Using Only Visible Beams	Select to determine the photometric value based on only instruments with **Draw Beam** selected; this reduces calculation time
Use Threshold Settings	Select to display the object in a color that corresponds to the specified threshold ranges
Lower Threshold	Specify the illumination value at the lower end of the range; values below this level are displayed in green, and values between the lower and upper threshold are displayed in tan
Upper Threshold	Specify the illumination value at the upper end of the range; values above this level are displayed in blue
Reset	Refreshes the photometer display after changes have been made to the lighting conditions

After placing the photometer, set the its Z value on the Object Info palette; illumination values vary depending on the photometer's elevation.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Obtaining Photometric Data
Inserting a Photometric Grid
Using Photometric Threshold Settings
Assigning a Focus Point to a Lighting Instrument
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

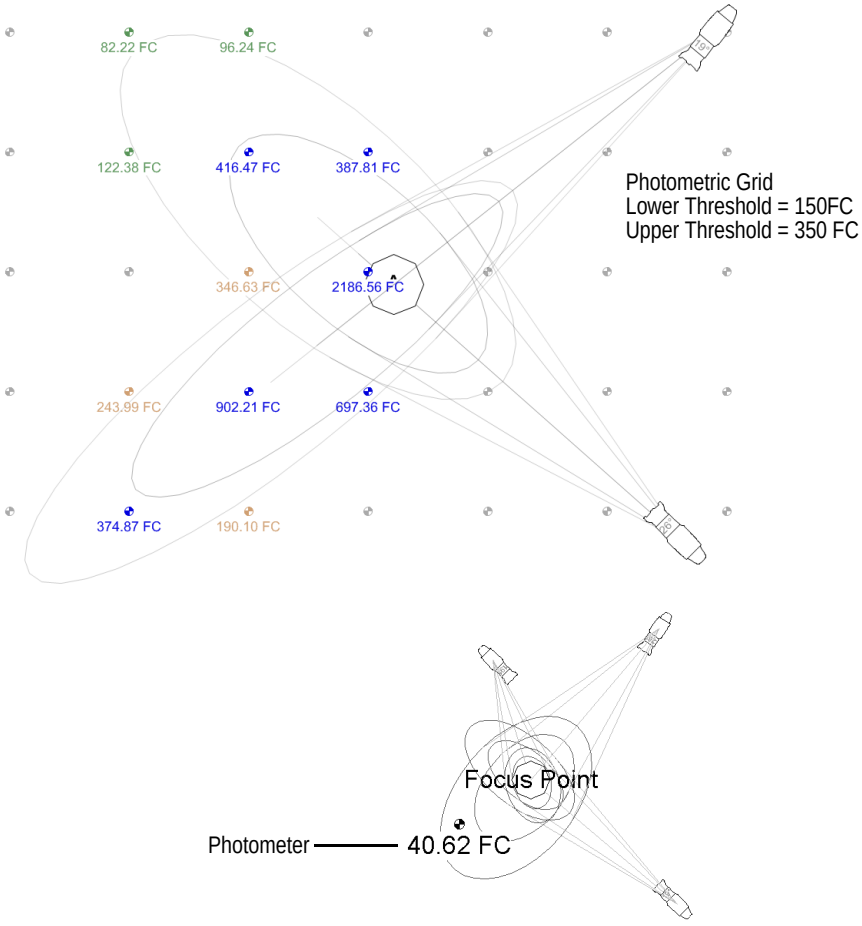
Using Photometric Threshold Settings

Vectorworks' photometric grid and photometer calculate and display the illumination values at the elevation point. The values are displayed in either foot candles (Imperial) or lux (Metric), depending on the units selected in **File > Document Settings > Units**.

If **Use Threshold Settings** is selected in the Object Info palette, colors indicate the illumination range according to the **Lower/Upper Threshold** values set in the Object Info palette.

Color	Description
Gray	Illumination value of zero
Green	Illumination value between zero and **Lower Threshold**
Tan	Illumination value between **Lower** and **Upper Threshold**
Blue	Illumination value above **Upper Threshold**

The photometric grid and photometer range colors can be changed. See "Customizing Photometric Threshold Colors" on page 904.



To evaluate the illumination values at different heights, change the elevation of the photometric object in the Object Info palette.

Customizing Photometric Threshold Colors

Customizing Photometric Threshold Colors

Both the photometer and photometric grid use colors that correspond to the specified threshold ranges (see “Obtaining Photometric Data” on page 899). These colors can be changed if desired.

To edit the photometric threshold color values:

1. Select **Tools > Plug-ins > Plug-in Manager**.

   The Plug-in Manager opens. Click on the Built-in Plug-ins tab and select either PhotoGrid or Photometer.

2. Click **Customize**.

   The Customize Plug-in dialog box opens. Click on the Strings tab.

3. Select Resource ID “4000: Threshold Color Indices.” Click **Edit**.

4. The Edit Strings dialog box opens. Select one of the indices and click **Edit** to specify a new color value for the threshold index.

ID	Corresponding Threshold Index
4000	Zero level
4001	Levels below the **Lower Threshold** value
4002	Levels between the **Lower** and **Upper Threshold** values
4003	Levels above the **Upper Threshold** value

   The values correspond to index values of the color palette. For more information on color palette selector values, see the Miscellaneous Selectors section of the developer-oriented documentation located at http://developer.vectorworks.net

5. When the desired threshold index color selector values have been edited, click **OK** to accept the changes, and then continue to click **OK** until you can close the Plug-In Manager dialog box.

Obtaining Photometric Data

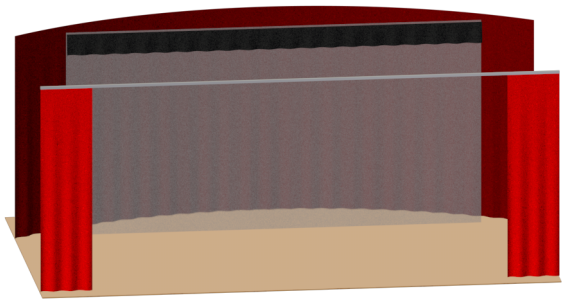
Inserting Soft Goods

The **Soft Goods** tool inserts theater and event planning draperies such as curtains, borders, scrims, and pipe-and-drape assemblies. Soft goods objects are drawn along a path, so they can be straight, curved, or in any configuration for total flexibility; the path can be edited after placement. The 3D appearance of soft goods objects can be set for a realistic appearance, or display more schematically. The materials used to create the soft goods configuration can be included in a report for cost estimation (see “Creating Reports” on page 1316). Text labels identify the components of the soft goods object.

Inserting Curtains and Borders
Inserting Pipe-and-Drape Assemblies
Setting Soft Goods 3D Display Options
Formatting Soft Goods Object Labels

Inserting Curtains and Borders

The **Soft Goods** tool creates a curved or straight stage curtain or border. To draw a curtain, either use the **Soft Goods** tool, or create a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).



To insert a curtain or border:

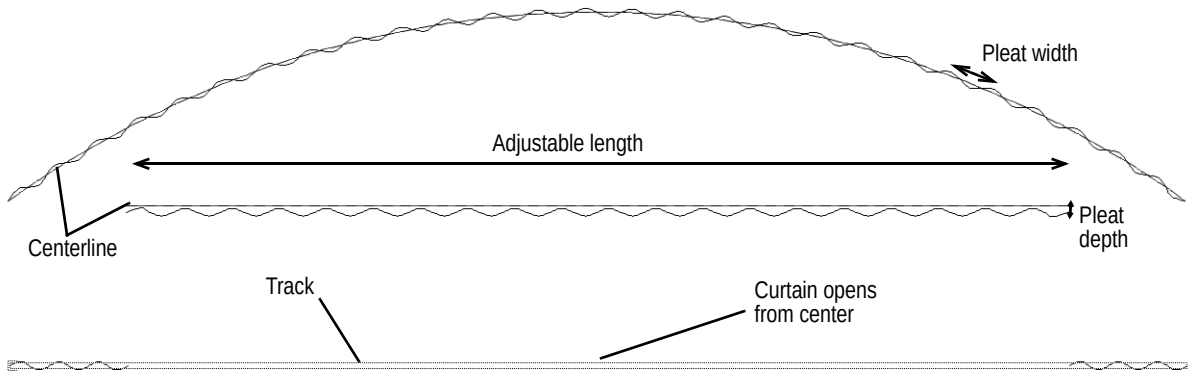
1. Click the **Soft Goods** tool from the appropriate tool set:
   - Spotlight workspace: Spotlight tool set
   - Designer workspace: Detailing tool set
2. Click **Preferences** from the Tool bar to specify or change any default **Soft Goods** tool parameters.

   The Soft Goods Object Properties dialog box opens.
3. Select either Curtain or Border from the **Function** list, and specify any other default parameters for this session.
4. Click **OK**.
5. Click on the appropriate mode in the Tool bar to select the creation method of the soft goods object.

   The Corner Vertex or Point on Arc modes are recommended. For more information on the **Polyline** tool modes, see “Creating Polylines” on page 298.
6. Click to set the soft goods object’s start point.
7. Click to set the end of the segment and the beginning of the next. Continue drawing segments in this manner until the curtain or border object is complete.

   Once created, the soft goods object can be reshaped by double-clicking on it. The **Reshape** tool is automatically activated, to reshape the object directly in the drawing.

   The curtain or border parameters can be edited in the Object Info palette. However, certain parameters are not available for borders.



Click to show/hide the parameters.

Parameter	Description
Function	Select either Curtain or Border (for pipe-and-drape assembles, see “Inserting Pipe-and-Drape Assemblies” on page 907)
Adjustable Length	Specifies the length of a straight curtain or border
Height	Indicates the height of the curtain or border
Show Centerline	In Top/Plan view, displays the curtain center line or the border reference line
Add Track	Draws the curtain track
Track Width	Sets the width of the curtain track
Show Traveler Pull	Shows the traveler pull on the left or right of the track, or hides the traveler pull
Open From	For curtains, creates an opening in the curtain at the center or at one of the ends. Select None to hide all openings in the curtain.
Width	When a curtain opening has been selected, specifies the width of the opening
Show Ghost Curtain	Displays the open section of the curtain with dotted lines in Top/Plan view
Reverse Side	For borders, switches the curtain to the other side of the center reference line
Running Length	For curved curtains, shows the total curtain length
Stock Drape Width	Specifies the width of an individual drape panel
Total Drape Count	Displays the number of drape panels required based on the specified stock drape width
Simple 2D	Temporarily simplifies the Top/Plan view of the curtain or border, showing it as a line
Simple 3D	Temporarily simplifies the 3D view of the curtain or border, showing it as a flat surface
Pleat Width	Indicates the distance between pleats
Pleat Depth	Indicates the depth of the curtain pleats
Flip Pleats	Switches the positive and negative portions of the pleats
Adjust Starting	For borders, offsets the starting point of the border
Adjust Ending	For borders, offsets the ending point of the border
3D Curtain Options	Opens the 3D Curtain Options dialog box, for setting the appearance of the curtain in 3D. An image (Renderworks required) or color can be set. To create a scrim, select Scrim from the **Soft Goods Options** list.

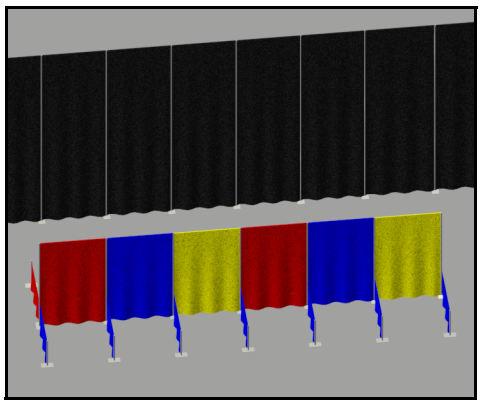
Parameter	Description
Current Option	Displays the currently selected 3D option
Location	Enter notes about the curtain or border location, if desired
Material Color	Enter notes about the curtain or border color (does not affect the curtain appearance)
Material Type	Enter notes about the curtain or border material (does not affect the curtain appearance)
Top Finish	Select the finish for the top of the curtain or border
Spacing	Sets the spacing for tie, hook, or grommet finishes
Bottom Finish	Select the finish for the bottom of the curtain or border
Sides Finish	Select the finish for the sides of the curtain or border
Fullness %	Specifies the curtain or border fullness information (does not affect curtain appearance)
Lining	Enter notes about the curtain or border lining material
Note	Enter any miscellaneous information required, such as stock number or other data
Class Soft Goods Parts	When first selected, automatically creates classes for the different parts of the curtain or border, for appearance and visibility control. This allows portions of the curtain or border to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected curtain or border.
Parts Classes Prefix	When soft goods parts are classed, creates a prefix for the class names so that they are sorted together
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Soft Goods Object Labels” on page 912
Default Text Positions	Text labels can be moved by dragging their control point; this option returns them to their default locations
Update	Updates the object when changes have been made to the Object Info palette parameters
Vertex Parameters	Edits the vertices of the path object that the curtain is based upon; see “Editing Vertex-Based Objects” on page 1002

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Pipe-and-Drape Assemblies
Setting Soft Goods 3D Display Options
Formatting Soft Goods Object Labels
Inserting Soft Goods

Inserting Pipe-and-Drape Assemblies

The **Soft Goods** tool creates pipe-and-drape assemblies typically used in event planning, such as when creating temporary booths in a convention center. To draw a pipe-and-drape assembly, either use the **Soft Goods** tool, or create a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



To insert a pipe-and-drape assembly:

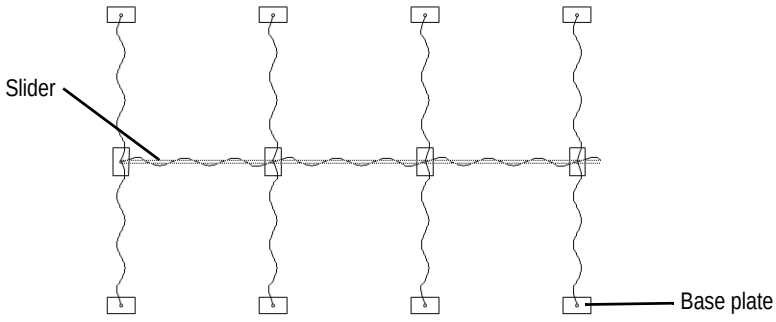
1. Click the **Soft Goods** tool from the appropriate tool set:
   - Spotlight workspace: Spotlight tool set
   - Designer workspace: Detailing tool set
2. Click **Preferences** from the Tool bar to specify or change any default **Soft Goods** tool parameters.

   The Soft Goods Object Properties dialog box opens.
3. Select Pipe-and-Drape from the **Function** list, and specify any other default parameters for this session.
4. Click **OK**.
5. Click on the appropriate mode in the Tool bar to select the creation method of the soft goods object.

   For more information on the **Polyline** tool modes, see “Creating Polylines” on page 298.
6. Click to set the soft goods object’s start point.
7. Click to set the end of the segment and the beginning of the next. Continue drawing segments in this manner until the pipe-and-drape assembly is complete.

   Once created, the soft goods object can be edited by selecting the object, and then selecting **Modify > Edit Soft Goods**. Reshape the soft goods polyline with the **Reshape** tool; click **Exit Profil**e to return to the drawing.

   The pipe-and-drape assembly parameters can be edited in the Object Info palette.



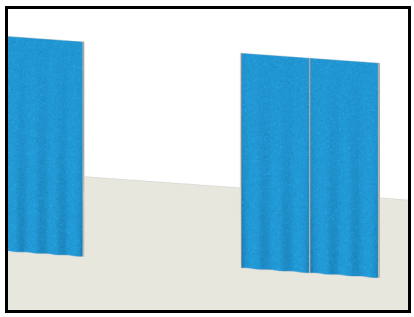
Click to show/hide the parameters.

Parameter	Description
Function	Select Pipe-and-Drape
Adjustable Length	Specifies the length of a straight pipe-and-drape assembly

Parameter	Description
Height	Indicates the height of the pipe-and-drape assembly
Show Centerline	In Top/Plan view, displays the center line of the pipe-and-drape assembly
Draw Slider(s)	Draws the soft goods slider segments at the top of the assembly
Slider Options	The number of sliders (panel segments) created depends on the **Adjustable Length** of the entire assembly, the selected **Slider Options** range, and the **Target Length** specified. Select from a list of standard slider size ranges, or select Custom to create a non-standard slider size and enter the number of panels to include.
Target Length	Sets the desired slider length within the range specified in **Slider Options**. If Custom was selected for the **Slider Options**, enter the specific length for the slider.
Actual Length	Displays the calculated slider length used for the segments
Slider Total	Displays the actual number of slider segments used; if Custom was selected for the **Slider Options**, specify the number of slider segments desired
Running Length	Displays the total length of the pipe-and-drape assembly
Base Plate Size	Select from a list of standard base plate sizes, or choose the rounded boom base
End Hardware	Select whether to include end hardware (base plate and upright) at each end of the assembly
First Plate Shift	When a base plate is included at the start of the assembly, select whether it is centered on the upright (None), or shifted towards the inside (In) or outside (Out). This option does not apply to a boom base.
Last Plate Shift	When a base plate is included at the end of the assembly, select whether it is centered on the upright (None), or shifted towards the inside (In) or outside (Out). This option does not apply to a boom base.
Upright/Plate Total	Displays the number of uprights and base plates in the assembly
Show 3D Uprights	Displays the upright hardware in 3D views; deselect this option for faster rendering
Stock Drape Width	Specifies the width of an individual drape panel
Total Drape Count	Displays the number of drape panels required based on the specified stock drape width
Simple 2D	Temporarily simplifies the Top/Plan view of the pipe-and-drape assembly, showing it as a line
Simple 3D	Temporarily simplifies the 3D view of the pipe-and-drape assembly, showing it as a flat surface
Pleat Width	Indicates the distance between pleats
Pleat Depth	Indicates the depth of the pleats
Flip Pleats	Switches the positive and negative portions of the pleats
3D Curtain Options	Opens the 3D Curtain Options dialog box, for setting the appearance of the pipe-and-drape assembly in 3D. An image (Renderworks required) or color can be set
Current Option	Displays the currently selected 3D option
Location	Enter notes about the pipe-and-drape assembly location, if desired
Material Color	Enter notes about the pipe-and-drape assembly color (does not affect the drapery appearance)

Parameter	Description
Material Type	Enter notes about the pipe-and-drape assembly material (does not affect the drapery appearance)
Note	Enter any miscellaneous information required, such as stock number or other data
Class Soft Goods Parts	When first selected, automatically creates classes for the different parts of the curtain or border, for appearance and visibility control. This allows portions of the curtain or border to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected curtain or border.
Parts Classes Prefix	When soft goods parts are classed, creates a prefix for the class names so that they are sorted together
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Soft Goods Object Labels” on page 912
Default Text Positions	Text labels can be moved by dragging their control point; this option returns them to their default locations
Update	Updates the object when changes have been made to the Object Info palette parameters
Vertex Parameters	Edits the vertices of the path object that the curtain is based upon; see “Editing Vertex-Based Objects” on page 1002.

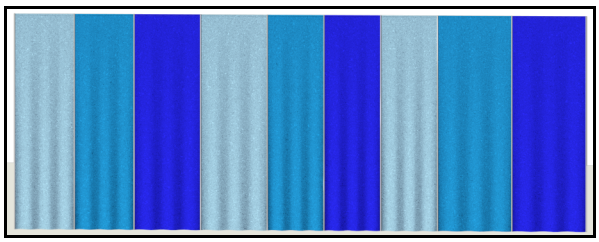
To create an opening in the pipe-and-drape assembly, edit the vertex parameters in the Object Info palette. Select the vertex prior to the opening as described in “Editing Vertex-Based Objects” on page 1002, and then click **Hide Next Edge**.



Inserting Curtains and Borders
Setting Soft Goods 3D Display Options
Formatting Soft Goods Object Labels
Inserting Soft Goods

Setting Soft Goods 3D Display Options

Select an image texture (Renderworks required) or color to display on the curtain, border, or pipe-and-drape assembly in 3D views. A curtain object can also be made to look like a scrim; pipe-and-drape assemblies can have alternating panel colors. The Vectorworks Spotlight product includes a number of sample image textures, or you can create your own textures (image textures or other textures; see “Creating Textures” on page 1505). The settings made here override any class settings made, if the soft goods object is assigned to a class; an alert dialog box displays if there is a conflict.



To set the 3D display options:

1. Select a curtain, border, or pipe-and-drape assembly (Vectorworks Spotlight required). From the Object Info palette, click **3D Curtain Options**.

   The 3D Curtain Options dialog box opens. Select the display from the **Soft Goods Options** list and then choose the formatting options.

Click to show/hide the parameters.

Parameter	Description
Soft Goods Options	Select the type of display for the soft goods material: • Opaque: Displays with a solid color • Scrim: Displays with a partly transparent color • Image (Renderworks required): Displays an image or assigned texture • Multi-Color: Displays each panel of the pipe-and-drape assembly with an alternating solid color
Opaque	
Opaque or Scrim Color	Specifies the color of the soft goods object in 3D views; click the color box to select the color
Scrim	
Opaque or Scrim Color	Specifies the color of the soft goods object in 3D views; click the color box to select the color
Image (Renderworks required)	
Image	Select the image texture to display on the soft goods object, from the current file's textures
Image Adjustments	If the selected image does not have the desired size or position in the Preview, adjust its scale or position
Scale	Increases or decreases the image scale
Horizontal Shift	Shifts the image horizontally to the left or right
Vertical Shift	Shifts the image vertically up or down
Flip Image	Flips the image
Preview	Displays the image selected for the soft goods object
Multi-Color (pipe-and-drape only)	

Parameter	Description
Multi-Color Properties	Lists the colors, in order, that will alternate on the pipe-and drape assembly panels To change the order of the colors, click in the # column and drag the color to a new position in the list.
Panel Color	Specifies a panel color; click the color box to select the color
Name	Enter a name for the specified color
Add Color	Adds the **Panel Color** to the **Multi-Color Properties** list, to be included among the panel colors. If a color has been changed, replaces the changed color in the list.
Remove Color	Removes the selected color from the **Multi-Color Properties** list

2. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Curtains and Borders
Inserting Pipe-and-Drape Assemblies
Formatting Soft Goods Object Labels

Formatting Soft Goods Object Labels

A variety of soft goods object labels can be included on the drawing.

To select labels for display and format the text:

1. Select a soft goods object. From the Object Info palette, click **Text Options**.

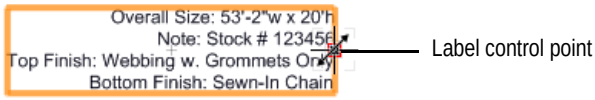
   The Text Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Optional Elements to Show	Select items to add to the text label; different label elements are available for curtains or borders and pipe-and-drape assemblies
Text Attributes	
Stack Text	Separates each selected label element by placing it in its own line of text; deselect this option to create a single line for all the selected elements
Automatically Position Text	Automatically places the label with the soft goods object; deselect this option to be able to reposition the label by dragging the label control point
Keep Text Horizontal	Maintains the text in a horizontal position even when the object is rotated; deselect this option to rotate the label along with the object
Include Element Labels	Includes the name of the element along with the value; deselect this option to display the parameter value only
Text formatting options	Specify the text size, color, and alignment

2. Specify the text label elements and formatting, and then click **OK**.

Once the label has been added to the drawing, it can be moved by clicking and dragging the label control point (unless **Automatically Position Text** was selected in the Text Options). Click **Default Text Positions** from the Object Info palette of a selected soft goods object to restore the text label to its original location.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Curtains and Borders
Inserting Pipe-and-Drape Assemblies
Setting Soft Goods 3D Display Options

Inserting Video Screen Objects

Video screen objects, such as televisions, projectors, and screens, are often a required part of visualizing a room layout for event planning, and are occasionally needed for theater productions. The Vectorworks Spotlight product automatically helps with the calculations required for placement, image size, and viewing area; it can also display a "glowing" image on the screen for a realistic look (Renderworks required). The video objects can be labeled with calculated information and certain objects display the optimum viewing area.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

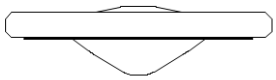
Inserting a Television Object
Inserting an LED Screen
Inserting a Video Screen Object
Inserting a Blended Screen and Projector
Setting the Image on the Video Screen
Formatting Video Screen Object Labels

Inserting a Television Object

The television object simulates CRT and flat screen televisions.

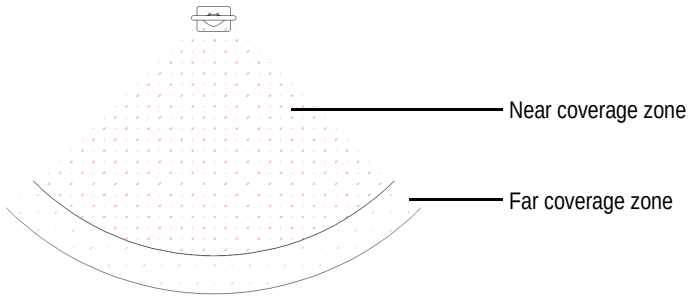
To insert a television:

1. Click the **Television** tool from the Spotlight tool set.
2. Click once in the drawing to set the object's position. Click again to set the object's rotation.
3. If this is the first time a television is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all televisions placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.



The television object's parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
TV Type	Select the type of TV: flat panel, CRT, or CRT curved
Screen Aspect	Select the aspect ratio of the screen: 4:3 (Standard) or 16:9 (Wide screen); curved CRT screens cannot have a 16:9 aspect ratio
Casing	Select the television model from the default content (see "Resource Libraries" on page 219); available models depend on the selected **TV Type** and **Screen Aspect**
Width (image)	Displays the width of the screen image area
Height (image)	Displays the height of the screen image area
Show Coverage Zone (2D only)	Indicates the viewing area based on the screen size; observers within the area should be able to see the screen 
Zone Reference	Calculates the coverage zone based on screen width, height, or diagonal measurement
Viewing Angle	Indicates the maximum viewing angle of the screen relative to dead-on
Near Multiplier	Specifies the multiplier of the reference dimension to determine the extent of the near coverage zone
Far Multiplier	Specifies the multiplier of the reference dimension to determine the extent of the far coverage zone
Edit Screen Image (Renderworks required)	Opens the Edit Screen Image dialog box to select the image for screen display.
Screen Image	Displays the name of the current screen image
Add Adjustable Stand	For flat panel televisions, adds a stand to the screen casing
Stand Height	Specifies the height measurement from the base of the stand to the bottom of the screen casing

Parameter	Description
Class Television Parts	When first selected, automatically creates classes for the different parts of the television and, if selected, stand, for appearance and visibility control. This allows portions of the television or stand to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected television.
Parts Classes Prefix	When screen parts are classed, creates a prefix for the class names so that they are sorted together
Note	Adds a note, which can be placed on the drawing with the **Text Options**
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Video Screen Object Labels” on page 929
Update	Updates the object when changes have been made to the Object Info palette parameters

Setting the Image on the Video Screen
Formatting Video Screen Object Labels

Inserting an LED Screen

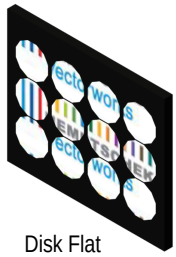
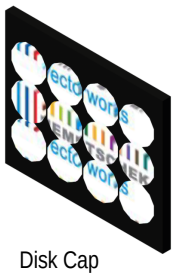
The Light-emitting Diode (LED) object simulates a variety of low-resolution LED screens in arrays on a base structure.

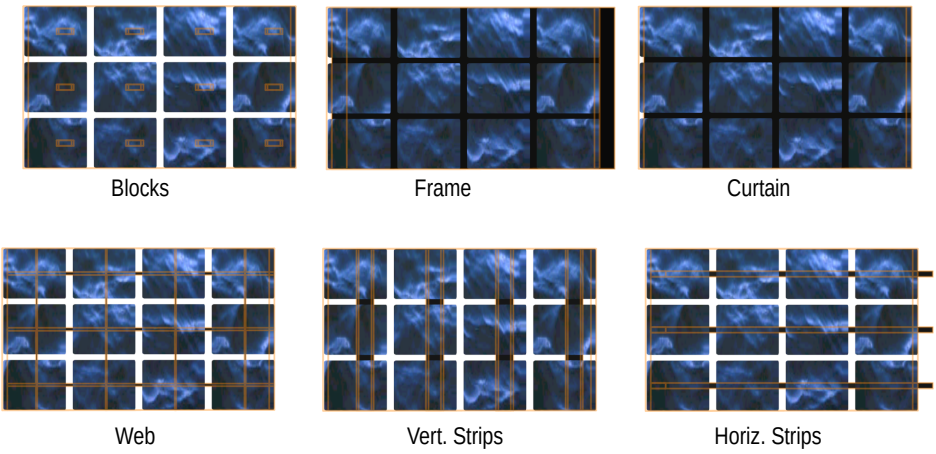
To insert an LED screen or array:

1. Click the **LED Screen** tool from the Spotlight tool set.
2. Click once in the drawing to set the object’s position. Click again to set the object’s rotation.
3. If this is the first time LED screens are placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all LED screens placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

The LED screen parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Module Shape	Select a shape for the screen modules in the array  Tile Flat  Tile Cap  Disk Flat  Disk Cap  Tube  Dome
Tile Flat	Creates rectangular-shaped modules
Tile Cap	Creates rectangular-shaped modules with frosted caps
Disk Flat	Creates round modules
Disk Cap	Creates round modules with frosted caps
Tube	Creates tube-shaped modules
Dome	Creates dome-shaped modules
Width	Specifies the width of each module in the array
Height	For tiled arrays, sets the height (length) of each module in the array
Cap Depth	For capped arrays, sets the depth of each module's cap
Tube Orientation	For tube arrays, specifies whether the tubes are oriented vertically or horizontally
Length	For tube arrays, sets the length of one of the tube modules
Build Array Based On	
Module Counts and Spacing	Sets the size of the entire LED array based on the number of modules specified in the horizontal and vertical directions, as well as the horizontal and vertical spacing distance set between the modules
Overall Dimensions and Spacing	Sets the size of the entire LED array based on the specified array width and height, as well as the horizontal and vertical spacing between the modules; the screen count is automatically calculated
Overall Dimensions and Module Counts	Sets the size of the entire LED array based on the specified array width and height, as will as the number of modules specified in the horizontal and vertical directions; the spacing is automatically calculated

Parameter	Description
Array Width/Height	Specifies the width and height of the entire LED array, when the array is based on overall dimensions
Horizontal/Vertical Spacing	Specifies the spacing between modules in the horizontal and vertical directions, when the array is based on spacing parameters
Horizontal/Vertical Count	Specifies the number of modules in the horizontal and vertical directions, when the array is based on the module counts
Module Total	Displays the total calculated number of modules in the array
Width/Height (Calc.)	Displays the calculated width and height of the array, when determined by module count and spacing
H/V Spacing (Calc.)	Displays the calculated module spacing in the horizontal and vertical directions, when the array is determined by the overall dimensions and module counts
H/V Count (Calc.)	Displays the total number of modules in the horizontal and vertical directions, when the array size is determined by the overall dimensions and module spacing
Pixels Per Module H/V	Specifies the number of horizontal or vertical pixels for each module (for calculation purposes only; does not affect the module display)
Array Size (Pixels)	Displays the total number of pixels for the entire LED array
Base Structure	Select a base structure for the array, or select None to display only the module(s); modify the appearance of the structure by selecting **Class LED Array Parts** and then editing the structure class  Blocks Frame Curtain Web Vert. Strips Horiz. Strips
Blocks	Creates a structure of individual blocks bases for each module
Frame	Creates a single frame for the entire array
Curtain	Creates a single background for the array
Web	Creates a webbed structure behind the modules
Vert. Strips	Places the LED modules in vertical columns
Horiz. Strips	Places the LED modules in horizontal rows
None	Does not create a base structure for the array; displays only the LED modules
Base Width	Specifies the width of blocks, vertical or horizontal strips, or the entire frame
Base Height	Specifies the height of blocks, vertical or horizontal strips, or the entire frame

Parameter	Description
Base Depth	Specifies the depth of blocks, vertical or horizontal strips, or the entire frame
Horizontal Inset	Sets the horizontal distance (margin) between the horizontal edge of the base structure and the array
Vertical Inset	Sets the vertical distance (margin) between the vertical edge of the base structure and the array
Pleat Width	For curtain or web structures, specifies the pleat width of the curtain (2D only for web structures)
Pleat Depth	For curtain or web structures, specifies the pleat depth of the curtain (2D only for web structures)
Webbing Width	For web structures, sets the width of the connecting web segments
Tilt	Specifies the tilt angle of the screen array
Edit Array Image (Renderworks required)	Opens the Edit Screen Image dialog box
Array Image	Displays the name of the current screen image
Simple 3D	Creates a single textured 3D polygon to represent the screen modules; this speeds rendering when each module does not need to be shown individually
Class LED Array Parts	When first selected, automatically creates classes for the different parts of the LED screen and, if selected, the base structure., for appearance and visibility control. This allows portions of the screen or base to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected LED array.
Parts Classes Prefix	When screen parts are classed, creates a prefix for the class names so that they are sorted together
Note	Adds a note, which can be placed on the drawing with the **Text Options**
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Video Screen Object Labels” on page 929
Default Text Positions	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Image on the Video Screen
Formatting Video Screen Object Labels

Inserting a Video Screen Object

Video screen objects consist of a video screen and optional front or rear projector.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To insert a video screen:

1. Click the **Video Screen** tool from the Spotlight tool set.
2. Click once in the drawing to set the object's position. Click again to set the object's rotation.
3. If this is the first time a video screen is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all video screens placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

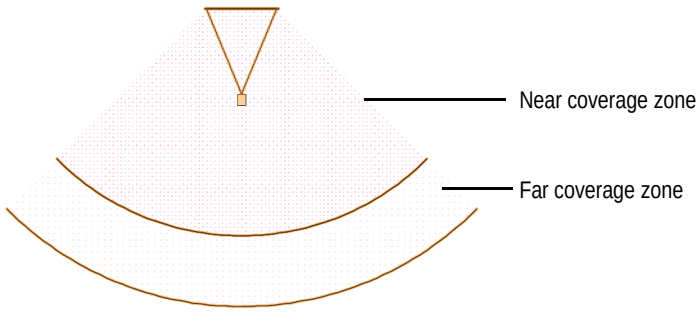
The height of the video screen and associated projector(s) depends on several factors.

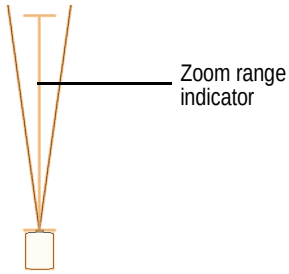
- The Z value entered in the Object Info palette determines the distance from the active layer plane to the bottom of the screen (including the border).
- When the screen includes legs, the legs are drawn on the active layer plane unless a **Floor Height** value has been specified. The **Floor Height** distance shifts the floor, and therefore the legs, by that amount from the layer plane.
- Projector stands are inserted relative to both the **Vertical Shift** and **Floor Height** values, allowing stands to be shifted up or down from the screen's floor as set by the **Floor Height** value.

The video screen object's parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Screen Type	Select the screen type: rear or front projector, or LED screen
Screen Aspect	Select one of the standard screen aspect ratios, or choose custom screen dimensions
Stock Size	Select one of the standard screen sizes based on height x width measurements
Width (image)	Displays the width of the screen image area (without the screen border or frame) for standard screen sizes. For custom screens, enter the screen width; the screen height is automatically calculated for all custom screens (except rect. all custom dimensions).
Height (image)	Displays the height of the screen image area (without the screen border or frame) for standard screen sizes. For custom screens, enter the screen height; the screen width is automatically calculated for all custom screens (except rect. all custom dimensions).
Diagonal (calc)	Displays the calculated diagonal screen size
Diameter (image)	For Round Custom screens, indicates the screen diameter
Frame	Select the type of border around the image area: No Frame, a Frame with a specified thickness, or a flat Border Only
Border Width(s) Top/Bottom/Left/ Right	For Frame and Border Only types, specify the width of the frame or border for each side of the frame or border

Parameter	Description
Depth	For Frame type, indicate the thickness of the frame
Edge	For Frame type, sets the distance between the screen and the front face of the screen's frame
Screen Tilt	When no **Screen Legs** are selected, specifies the tilt of the screen in degrees. Positive values tilt the top of the screen away from the viewer, and negative values tilt it toward the viewer.
Total Clearance	Displays the total vertical clearance height required by the complete video screen object from the floor to the top of the screen, including any border, dress kit, valence, or other accessories
Show Coverage Zone (2D only)	Indicates the viewing area based on the screen size; observers within the area should be able to see the screen 
Zone Reference	Calculates the coverage zone based on screen width, height, or diagonal measurement
Viewing Angle	Indicates the maximum viewing angle of the screen relative to dead-on
Near Multiplier	Specifies the multiplier of the reference dimension to determine the extent of the near coverage zone
Far Multiplier	Specifies the multiplier of the reference dimension to determine the extent of the far coverage zone
Screen Support	Select the type of screen support: None, Goalpost (includes two legs, and is similar to commercial "fastfold" systems), Tripod (creates a "roll-up" type support arrangement), Roll-Down Case, or Roll-Up Case
Show Feet	For Goalpost configurations, select how the feet of the frame display: Both, Front, Back, or None
Depth	For case supports, specifies the size of the case drawn at the top (roll-down case) or bottom (roll-up case) of the video screen; the case length is determined by the screen size
Motor	For case supports, adds space to the case for the motor; select whether the motor is located on the right or the left, or select None for manually operated screens
Opening	For case supports, sets the location on the case where the screen exits the case; select Front, Center, or Back
Add Dress Kit	When Goalpost or Tripod screen legs are selected, adds a "dress kit" of draperies to conceal the projector (rear) and supports from audience view
Dress Kit Color	Select the color of the dress kit draperies
Left Leg Width	Indicates the width of the drapery on the left side of the screen

Parameter	Description
Right Leg Width	Indicates the width of the drapery on the right side of the screen
Valence Height	Sets the height of the drapery above the screen
Border Overlap	Indicates the amount of overlap of the dress kit drapery over all sides of the screen's border
Pleat Width	Specifies the width of each pleated section of the left, right, and valence draperies
Pleat Depth	Specifies the depth of the pleats for the left, right, and valence draperies
Hide Screen	Toggles the display of the screen geometry
Edit Screen Image (Renderworks required)	Opens the Edit Screen Image dialog box
Screen Image	Displays the name of the current screen image
Show Projector	Toggles the display of the projector unit
Show Projection Cone	Toggles the display of the projection cone
Projector Model	Select the projector model from the default content (see "Resource Libraries" on page 219)
Projector Aspect	For screens with a portrait orientation, select a standard projector aspect ratio
L/R Shift	Shifts the projector to the left or right relative to the screen
Horiz. Offset Angle	Displays the angle of offset from the projector to the screen, measured side to side on the horizontal plane and based on the **L/R Shift** value
Horiz. Offset Perc.	Displays the angle of offset from the projector to the screen as a percentage
Point at Screen Center	When the projector has been shifted to the left or right, keeps the projector pointed at the center of the screen
Projector Tilt	Sets the tilt of the projector, in degrees, relative to the horizontal plane
Place Based On	Sets the position of the projector either based on a fixed lens size or the projection distance
Projection Dist.	When Distance is selected for the projector placement, enter a distance value or click and drag the projectors on the drawing to set the distance
Lens	Select a standard fixed or zoom lens size, or choose Custom. When the projector is placed based on distance, this displays the calculated lens size.
Zoom Factor	For zoom or custom lenses, sets the lens zoom factor
Show Zoom Range	For zoom lenses, indicates the optimal area on the drawing where the projector should be placed, to achieve the desired image size within the zoom range of the lens 

Parameter	Description
Vertical Position	Select the projector placement mode: • Screen Center: The center of the projector lens aligns with the screen center • Align to Top: The top of the projector body aligns with the top of the top of the screen border • Align to Bottom: The bottom of the projector body aligns with the bottom of the screen border • Stand: Places the projector on a stand selected in **Stand Model** • Rigged: Places the projector at a height specified in **Floor Height** with the bottom of the projector at the **Trim** height • Specific Shift: Shifts the projector relative to the screen center by the **Vertical Shift** distance, as measured from the center of the projector lens
Vertical Shift	For projectors on a stand or with a specific shift, specifies the distance between the floor (as set by the **Floor Height**) and the stand or projector (this allows stands or projectors to be placed on a plane shifted up or down from the screen)
Stand Model	For projectors on a stand, select the projector stand model from the default content (see "Resource Libraries" on page 219)
Floor Height	For rigged projectors or projectors on a stand, indicates the distance from the active layer plane to the floor, effectively shifting the floor by the indicated height
L/R Shift	For projectors on a stand, shifts the stand to the left or right, relative to the projector
F/B Shift	For projectors on a stand, shifts the stand to the front or back, relative to the projector
Trim (bottom)	For rigged projectors, indicates the location of the bottom of the projector
Vert. Offset Angle	Displays the vertical offset angle of the projector to the screen, based on the vertical position of the projector
Vert. Offset Perc.	Displays the vertical offset angle of the projector to the screen as a percentage
Multiple Projector	When there is more than one projector, select whether they are stacked or side-by-side
Horiz Space	For side-by-side multiple projectors, sets the distance between the projectors
Offset Distance	For side-by-side multiple projectors, sets the offset distance between the screen and the second projector, relative to the first projector
Class Video Screen Parts	When first selected, automatically creates classes for the different parts of the video screen for appearance and visibility control. This allows portions of the video screen, projector, and other elements to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected video screen object.
Parts Classes Prefix	When screen parts are classed, creates a prefix for the class names so that they are sorted together
Note	Adds a note, which can be placed on the drawing with the **Text Options**
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see "Formatting Video Screen Object Labels" on page 929
Default Text Positions	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Image on the Video Screen
Formatting Video Screen Object Labels

Inserting a Blended Screen and Projector

The blended screen simulates projection screens that require multiple projectors to produce one large image.

To insert a blended screen and projector(s):

1. Click the **Blended Screen** tool from the Spotlight tool set.
2. Click once in the drawing to set the object's position. Click again to set the object's rotation.
3. If this is the first time a blended screen is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all blended screens placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

When you insert a blended screen, a Blended Screen object is created. Once the parameters of the blended screen and projectors have been set, click **Insert Projectors** from the Object Info palette to create the projectors associated with that screen. The blended screen settings control the initial projector settings, though each projector can then be set independently (by changing parameters like text position or cone display) if needed. Moving or rotating a blended screen after the projectors have been inserted also moves or rotates the associated projectors.

> Use caution when clicking **Insert Projectors** if projectors have already been inserted; this deletes existing projectors and their parameter settings.

The height of the blended screen and associated projectors depends on several factors.

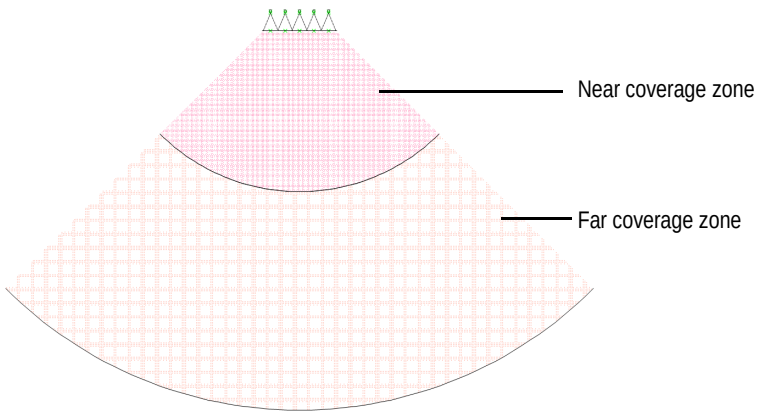
- The Z value entered in the Object Info palette determines the distance from the active layer plane to the bottom of the screen (including the border).
- When the screen includes legs, the legs are drawn on the active layer plane unless a **Floor Height** value has been specified. The **Floor Height** distance shifts the floor, and therefore the legs, by that amount from the layer plane.
- Projector stands are inserted relative to both the **Vertical Shift** and **Floor Height** values, allowing stands to be shifted up or down from the screen's floor as set by the **Floor Height** value.

The blended screen and blended projector object parameters can be edited in the Object Info palette.

Blended Screen Parameters

Click to show/hide the parameters.

Parameter	Description
Screen Type	Select either front or rear projector
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Screen Name	Provide a name for the screen object; associated blended projectors identify the screen by this name; if the name is already in use, the name that was already in use changes automatically
Width (image)	Enter the width of the screen image area (without the screen border or frame)
Height (image)	Enter the height of the screen image area (without the screen border or frame)
Screen Size (pixels)	Displays the total screen size based on the screen dimensions
Frame	Select the type of border around the image area: No Frame, a Frame with a specified thickness, or a flat Border Only
Border Widths Top/Bottom/Left/ Right	For Frame and Border Only types, specify the width of the frame or border for each side of the frame or border
Depth	For Frame type, indicate the thickness of the frame
Edge	For Frame type, sets the distance between the screen and the front face of the screen's frame
Screen Tilt	When **Add Folding Legs** is selected, specifies the tilt of the screen in degrees. Positive values tilt the top of the screen away from the viewer, and negative values tilt it toward the viewer.
Total Clearance	Displays the total vertical clearance height required by the complete blended screen object from the floor to the top of the screen, including any border, dress kit, valence, or other accessories
Screen Aspect (calc)	Displays the aspect ratio of the screen
Show Coverage Zone (2D only)	Indicates the viewing area based on the screen size; observers within the area should be able to see the screen 
Zone Reference	Calculates the coverage zone based on screen width, height, or diagonal measurement
Viewing Angle	Indicates the maximum viewing angle of the screen relative to dead-on
Near Multiplier	Specifies the multiplier of the reference dimension to determine the extent of the near coverage zone
Far Multiplier	Specifies the multiplier of the reference dimension to determine the extent of the far coverage zone

Parameter	Description
Screen Support	Select the type of screen support: None, Goalpost Legs (includes two legs, and is similar to commercial “fastfold” systems), or Roll-Down Case
Show Feet	For Goalpost Legs, select how the feet of the frame display: Both, Front, Back, or None
Depth	For Roll-Down Case support, specifies the size of the case drawn at the top (roll-down case) or bottom (roll-up case) of the video screen; the case length is determined by the screen size
Motor	For Roll-Down Case support, adds space to the case for the motor; select whether the motor is located on the right or the left, or select None for manually operated screens
Opening	For Roll-Down Case support, sets the location on the case where the screen exits the case; select Front, Center, or Back
Add Dress Kit	When goalpost legs are selected, adds a “dress kit” of draperies to conceal the projector (rear) and supports from audience view
Dress Kit Color	Select the color of the dress kit draperies
Left Leg Width	Indicates the width of the drapery on the left side of the screen
Right Leg Width	Indicates the width of the drapery on the right side of the screen
Valence Height	Sets the height of the drapery above the screen
Border Overlap	Indicates the amount of overlap of the dress kit drapery over all sides of the screen’s border
Pleat Width	Specifies the width of each pleated section of the left, right, and valence draperies
Pleat Depth	Specifies the depth of the pleats for the left, right, and valence draperies
Edit Screen Image (Renderworks required)	Opens the Edit Screen Image dialog box
Screen Image	Displays the name of the current screen image
Projector Model	Select the projector model from the default content (see “Resource Libraries” on page 219); specify the projector settings, which apply to all projectors upon insertion
Projector Aspect	Select the projector aspect ratio
Proj. Resolution	Select the projector pixel resolution
Overshoot %	Specifies the total amount of top and bottom overshoot to be discarded when using narrow aspect projectors to achieve a certain pixel blend
Projector Tilt	Sets the tilt of the projector, in degrees, relative to the horizontal plane
Place Based On	Sets the position of the projector either based on a fixed lens size or the projection distance
Projection Dist.	When Distance is selected for the projector placement, enter a distance value or click and drag the projectors on the drawing to set the distance
Lens	Select a standard fixed or zoom lens size, or choose Custom. When the projector is placed based on distance, this displays the calculated lens size.
Zoom Factor	For zoom or custom lenses, sets the lens zoom factor

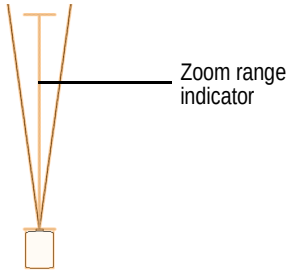
Parameter	Description
Vertical Position	Select the projector placement mode: • Screen Center: The center of the projector lens aligns with the screen center • Align to Top: The top of the projector body aligns with the top of the top of the screen border • Align to Bottom: The bottom of the projector body aligns with the bottom of the screen border • Stand: Places the projector on a stand selected in **Stand Model** • Rigged: Places the projector at a height specified in **Floor Height** with the bottom of the projector at the **Trim** height • Specific Shift: Shifts the projector relative to the screen center by the **Vertical Shift** distance, as measured from the center of the projector lens
Vertical Shift	For projectors on a stand or with a specific shift, specifies the distance between the floor (as set by the **Floor Height**) and the stand or projector (this allows stands or projectors to be placed on a plane shifted up or down from the screen)
Stand Model	For projectors on a stand, select the projector stand model from the default content (see “Resource Libraries” on page 219)
Floor Height	For rigged projectors or projectors on a stand, indicates the distance from the active layer plane to the floor, effectively shifting the floor by the indicated height
Trim (bottom)	For rigged projectors, indicates the location of the bottom of the projector
Multiple Projector	When there is more than one projector (identical paired projectors for each section of the screen), select whether they are stacked or side-by-side
Horiz Space	For side-by-side multiple projectors, sets the distance between the projectors
Area Horiz. Count	Sets the number of areas into which the screen is divided; if the value entered is too small, the lowest possible number of areas is automatically set
Area Width	Displays the width of each screen area
Area Height	Displays the height of each screen area
Proj. Spacing	Displays the distance between projectors, measured from the center of each projector lens
Overlap (meas.)	Displays the amount of screen area overlap, measured in distance units
Overlap (pixels)	Displays the amount of screen area overlap, measured in pixels
Overlap (%)	Displays the amount of screen area overlap, as a percentage
Show Projection Cones	Toggles the display of the projection cone for all projectors associated with the screen
Class Blended Screen Parts	When first selected, automatically creates classes for the different parts of the blended screen for appearance and visibility control. This allows portions of the screen, projector, and other elements to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected video screen object.
Parts Classes Prefix	When screen parts are classed, creates a prefix for the class names so that they are sorted together

Parameter	Description
Note	Adds a note, which can be placed on the drawing with the **Text Options**
Insert Projectors	Inserts the required number of blended projectors based on the specified parameters; these projectors are associated with the blended screen. The projector model specified in **Projector Model**, and its associated parameters, apply to all inserted projectors.
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Video Screen Object Labels” on page 929
Default Text Positions	Restores text labels to their default positions
Update	Updates the blended screen object and its associated projectors when changes have been made to the screen’s Object Info palette parameters; this also refreshes the associated projector information without changing individual blended projector parameters such as text options

Blended Projector Parameters

Click to show/hide the parameters.

Parameter	Description
Screen Name	Displays the name of the screen with which the projector is associated
Screen Parameters	Displays the main screen parameters set by the blended screen
Show Projection Cone	Toggles the display of the projection cone for this projector only
Projector Model	Select the projector model from the default content (see “Resource Libraries” on page 219); by default, the projector model specified for the blended screen is selected, but can be overridden for individual blended projectors
Show Centerline (Area)	Displays the projector centerline in Top/Plan view
L/R Shift	For projectors on a stand, shifts the stand to the left or right, relative to the projector
F/B Shift	For projectors on a stand, shifts the stand to the front or back, relative to the projector
Horiz. Offset Angle	Displays the angle of offset from the projector to the screen, measured side to side on the horizontal plane and based on the **L/R Shift** value
Horiz. Offset Perc.	Displays the angle of offset from the projector to the screen as a percentage
Point at Screen Center	When a L/R shift has been applied to the projector, rotates the projector body so that the projection remains pointed at the center of the projection cone area
Tilt	Sets the tilt of the projector, in degrees, relative to the horizontal plane
Place Based On	Sets the position of the projector either based on a fixed lens size or the projection distance
Projection Dist.	When Distance is selected for the projector placement, enter a distance value or click and drag the projectors on the drawing to set the distance
Lens	Select a standard fixed or zoom lens size, or choose Custom. When the projector is placed based on distance, this displays the calculated lens size.
Zoom Factor	For zoom or custom lenses, sets the lens zoom factor

Parameter	Description
Show Zoom Range	For zoom lenses, indicates the optimal area on the drawing where the projector should be placed, to achieve the desired image size within the zoom range of the lens 
Vertical Position	Select the projector placement mode: • Screen Center: The center of the projector lens aligns with the screen center • Align to Top: The top of the projector body aligns with the top of the top of the screen border • Align to Bottom: The bottom of the projector body aligns with the bottom of the screen border • Stand: Places the projector on a stand selected in **Stand Model** • Rigged: Places the projector at a height specified in **Floor Height** with the bottom of the projector at the **Trim** height • Specific Shift: Shifts the projector relative to the screen center by the **Vertical Shift** distance, as measured from the center of the projector lens
Vertical Shift	For projectors on a stand or with a specific shift, specifies the distance between the floor (as set by the **Floor Height**) and the stand or projector (this allows stands or projectors to be placed on a plane shifted up or down from the screen)
Stand Model	For projectors on a stand, select the projector stand model from the default content (see “Resource Libraries” on page 219)
Floor Height	For rigged projectors or projectors on a stand, indicates the distance from the active layer plane to the floor, effectively shifting the floor by the indicated height
Trim (bottom)	For rigged projectors, indicates the location of the bottom of the projector
Vert. Offset Angle	Displays the vertical offset angle of the projector to the screen, based on the vertical position of the projector
Vert. Offset Perc.	Displays the vertical offset angle of the projector to the screen as a percentage
Multiple Projector	When there is more than one projector (identical paired projectors for each section of the screen), select whether they are stacked or side-by-side
Horiz Space	For side-by-side multiple projectors, sets the distance between the projectors
Note	Adds a note, which can be placed on the drawing with the **Text Options**
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Video Screen Object Labels” on page 929
Default Text Positions	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

Setting the Image on the Video Screen
Formatting Video Screen Object Labels

S R Setting the Image on the Video Screen

Select an image to display on the video screen or LED screen (Renderworks required). The Vectorworks Spotlight product includes a number of sample images, or you can create your own textures. Textures must use an image color shader with constant reflectivity to display on the video screen (see “Creating Textures” on page 1505).

To set the video screen image:

1. Select a video screen object, blended screen, or LED screen object. From the Object Info palette, click **Edit Screen Image** or **Edit Array Image**.

   The Edit Screen Image or Edit Array Image dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Screen Image	Select the screen image to display on the television, LED, or video screen from either the default content (see “Resource Libraries” on page 219) or the current file’s textures.
Image Adjustments	If the selected image does not have the desired size or position in the Preview, adjust its scale or position
Scale	Increases or decreases the image scale
Horizontal Shift	Shifts the image horizontally to the left or right
Vertical Shift	Shifts the image vertically up or down
Tile Image	Repeats the image over the screen; deselect to show only a single instance of the image; does not apply to LED screens
Preview	Displays the screen image at the correct aspect ratio for the video screen

2. If needed, adjust the image scale, and shift the image horizontally or vertically until it appears correctly in the preview, and then click **OK**.

Inserting Video Screen Objects
Inserting an LED Screen
Formatting Video Screen Object Labels

S Formatting Video Screen Object Labels

A variety of video screen object labels can be included on the drawing. Different options are available depending on the video screen object.

To select labels for display and format the text:

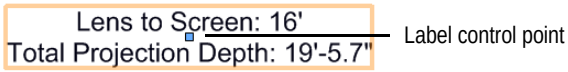
1. Select a video screen object. From the Object Info palette, click **Text Options**.

   The Text Options dialog box opens.

Text Option	Description
Coverage Zone	Displays the near and far coverage zone distances and multipliers at the specified text size and color
Image Dimensions	Displays the screen image dimensions, and if selected, the image size, screen clearance, and aspect ratio values, at the specified text size, color, and alignment. If desired, the image dimension text order can be changed from the default height x width to width x height instead, by selecting **Swap Dimension Order**.
Lens Information	Displays the projector lens size at the specified text size, color, and alignment
Projection Distance	Displays the distance from the projector to the screen at the specified text size, color, and alignment. Select the options to show the straight (plan) distance in 2D, direct distance (projector throw distance) in 2D, and/or actual distance in 3D. A dimension line can be included; specify the marker size.
Projector IDs	Labels the projector body with the entered ID information (such as projector name), at the specified text size and color. Any Note text is not displayed, but stores internal information about the projector.
Multiple Projector Tag	Displays the multiple projector layout for stacked projectors at the specified text size and color
Show Array Dimensions	For LED screen arrays, displays the overall array dimensions and total number of modules at the specified text size, color, and alignment
Note	Show the text entered for **Note** in the Object Info palette of the video screen object, at the specified text size, color, and alignment. A **Note Label** prefix can also be included; enter the contents of the prefix.
Keep Text Horizontal	Maintains the text in a horizontal position even when the object is rotated; deselect this option to rotate the label along with the object
Fill Text Background	Allows a text background fill to be used for all video screen text; by default, white is used as the fill color. To specify a different fill color, select the Object Info palette option to class the parts of the video screen object, and then select a fill color for the class of the video screen text. The class should be set to **Use at Creation**.

2. Specify the text labels and formatting, and then click **OK**.

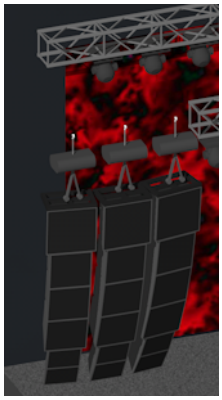
Once the labels have been added to the drawing, they can be moved by clicking and dragging the label control point. Click **Default Text Positions** from the Object Info palette of a selected video object to restore the text labels to their original locations.



Inserting Video Screen Objects

Inserting Speakers and Speaker Arrays

Audio objects, including speakers and arrays of speakers, are often a required part of visualizing a room or stage layout for event planning and entertainment design, and can also be needed for theater productions. Vectorworks creates speaker system layout drawings and can perform basic audio coverage analysis. The speakers and speaker arrays can be labeled with information and calculated data and can display the optimum listening area for up to three ranges.



Speaker arrays

Speaker

The Vectorworks Spotlight software comes with default speaker data located in the audio tools folders of the [Vectorworks]\Libraries folder that is included with the Vectorworks Spotlight product (see “Resource Libraries” on page 219). You can also save speaker data to a library file, which can be shared with others, and you can import speaker data from other files. The file(s) must be located within the Audio Tools\Speakers or Audio Tools\Bumpers folders.

Another way to share speakers or speaker arrays is to create symbols from them and import the symbol into another file.

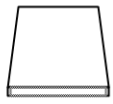
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

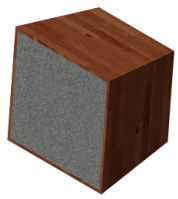
Inserting Speakers

Individual speakers that are not part of an array are inserted with the **Speaker** tool. Manually-inserted speakers can be placed as single speakers, or as several speakers arranged in a column. A variety of supports and labeling options are available. Use a speaker type and information with pre-set data from the speaker library file located in the [Vectorworks]\Libraries folder that is included with the Vectorworks Spotlight product (see “Resource Libraries” on page 219), or create a speaker with custom data and save it to the library for future use and sharing.

To insert a speaker:

1. Click the **Speaker** tool from the Spotlight tool set.
2. Click once in the drawing to set the object’s position. Click again to set the object’s rotation.
3. If this is the first time a speaker is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all speakers placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

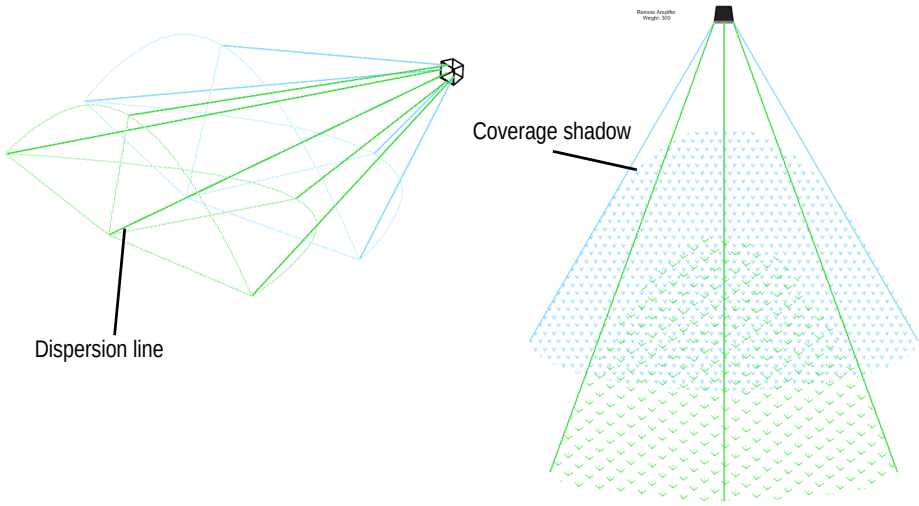




The speaker object’s parameters can be edited in the Object Info palette.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Type	Select a speaker from the default speaker data available in the library file; alternatively, select <Generic> to create a speaker with custom data and then save the speaker as a custom speaker in the library file. Select **Import** to import predefined speaker data only (such as width and weight) from a default content library file or other file containing speakers. The Import Speaker Type dialog box opens to select the file and type. A symbol is not imported from the file; only data is added.
Brand	Specifies the speaker manufacturer
Model	Specifies the manufacturer's information
Front Width/Height	Indicates the width and height of the front face of the speaker
Back Width/Height	Indicates the width and height of the back face of the speaker; the back face cannot be larger than the front face
Depth	Specifies the speaker depth
Flip Orientation	Rotates the speaker 90° to place the speaker on its side
Tilt Reference	When the speaker is in an array, sets the reference for tilting the speaker to either the front or back face
Weight	Indicates the weight of the speaker
Self-Powered	Indicates whether the speaker uses an internal or remote amplifier
Expand Dispersion Features	When selected, allows you to enter range notes and dispersion area information for up to three ranges; deselect to enter only Overall information
Range 1/2/3	Enter notes for three dispersion ranges, such as a frequency range, key frequency, or crossover point. When **Expand Dispersion Features** is deselected, the range is automatically set to Overall and only one set of notes can be entered.
Horiz./Vert. Dispersion	Enter the horizontal and vertical dispersion angle(s), from 0 to 180°; ranges 2 and 3 are only available when **Expand Dispersion Features** is selected
Throw Distance	Sets the throw distance reference for the dispersion lines; ranges 2 and 3 are only available when **Expand Dispersion Features** is selected
Save Type to Library	Saves the current speaker configuration as a new speaker type; enter the name of the speaker type. This custom speaker can then be selected from the **Type** list.
Delete Type from Library	Removes the speaker currently selected in **Type** from the library file; the current parameters remain as set, and the type becomes <Generic>
Listening Height	Sets the listening height reference for the dispersion lines. Listening height must be less than the speaker's elevation (Z value).

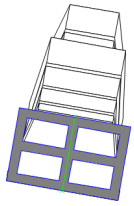
Parameter	Description
Show Dispersion Range 1/2/3	Draws the top, bottom, left, and right dispersion lines, along with a center reference line, for each of the three dispersion ranges selected for display (or the Overall range if expanded dispersion is not selected). An outline or hatch pattern representing the listening area coverage "shadow" is drawn when the throw distance and the vertical dispersion and tilt angles allow the speaker to cover the area at or below the **Listening Height**. 
Show Dispersion Hatch	Displays hatch fills in the coverage shadow of each enabled dispersion range
Support	Select the support method for the speaker. The options are for reference only, except for Tripod, which creates a tripod floor stand in the drawing, and Ground-Stacked, which allows speakers in an array to be ground-stacked rather than flown.
Stand Height	For tripod support, sets the height of the floor stand
Tilt Difference	Displays the tilt difference between the selected speaker and the one above it
Actual Tilt Angle	Specifies the tilt angle of the speaker, from -90 to 90°
Location	Indicates the speaker location (such as Downstage Right)
Purpose	Indicates the purpose of the speaker (such as Center Fill)
Rack	Enter notes about the rack driving the speaker
Drive Line	Indicates information about the drive line feeding the signal to the speaker
Column ID	An individual speaker can optionally be part of a column of speakers or in a speaker array. This specifies the column ID. Speakers in the same column receive a stacked Column ID when **Arrange Column** is clicked.
Position in Column	When speakers are arranged in a column or array, indicates the speaker's position in the stack. Position 1 is the top speaker.
Arrange Column	Provides selected speakers with a **Column ID** in order of their arrangement, based on their elevation (Z value); calculates tilt differences, and assigns each speaker a **Position in Column** number
Notes	Enter miscellaneous notes such as rental source or stock number

Parameter	Description
Class Speaker Parts	When first selected, automatically creates classes for the different parts of the speaker, for appearance and visibility control. This allows portions of the cabinets, grills, and other elements to be set to visible, grayed, or invisible. Different textures can be applied to the different part classes (Renderworks required), for a highly realistic appearance. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected speaker object. You may need to click **Update** to view any changes.
Parts Classes Prefix	When speaker parts are classed, enter a prefix for the class names so that they are sorted together
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Speaker Object Labels” on page 939
Default Text Position	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters. For example, you may need to refresh the display when toggling **Class Speaker Parts** on and off, when the attributes of the speaker parts have been set by class.

Inserting Speakers and Speaker Arrays
Formatting Speaker Object Labels
Inserting Speaker Arrays

Inserting Speaker Arrays

Large venues require the use of speaker arrays, which consist of speakers stacked in a column and topped by a bumper (top mounting bracket). Up to three different types of speakers can be included in an array. The speakers can each be tilted differently to provide maximum audio coverage for the audience.

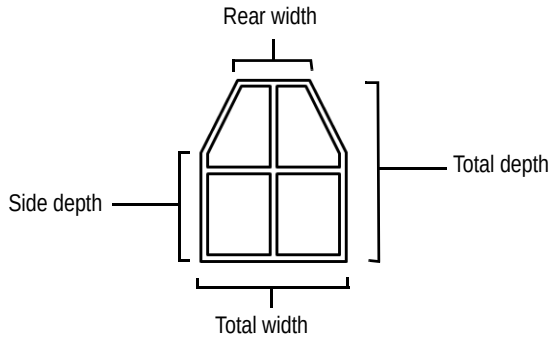


To insert a speaker array:

1. Click the **Speaker Array** tool from the Spotlight tool set.
2. Click once in the drawing to set the object’s position. Click again to set the object’s rotation.
3. If this is the first time a speaker array is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all arrays placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.
4. A bumper is inserted on the drawing. From the Object Info palette, click **Configure Array**.

   The Array Detail and Configuration dialog box opens. Specify the bumper parameters on the Bumper tab.

Click to show/hide the parameters.

Parameter	Description
Definition	
Type	Select a bumper from the default bumper data available in the library file; alternatively, select <Generic> to create a bumper with custom data and then save the bumper as a custom bumper in the library file. Select Import to import predefined bumper data only (such as width and weight) from a default content library file or other file containing bumpers. The Import Bumper Type dialog box opens to select the file and type. A symbol is not imported from the file; only data is added. Select None to have no bumper included in the speaker array, such as for ground-stacked speakers or speakers with built-in hardware for installation.
Brand	Specifies the bumper manufacturer
Model	Specifies the manufacturer's information
Save Type to Library	Saves the current bumper configuration as a new bumper type; enter the name of the bumper type. This custom bumper can then be selected from the **Type** list.
Delete Type from Library	Removes the speaker currently selected in **Type** from the library file; the current parameters remain as set, and the type becomes <Generic>
Attributes	
Total Width	Indicates the width of the entire bumper
Rear Width	Sets the width of the bumper at the back of the bumper
Total Depth	Specifies the total depth of the bumper
Side Depth	Sets the depth of the bumper at the side
Thickness	Specifies the thickness of the bumper
Hardware Width	Sets the width of the bumper hardware (brackets)
Weight	Specifies the bumper weight
Preview	Displays a preview of the bumper, which updates as parameters change; the arrow indicates the front face

5. On the Speaker A, B, and C tabs, configure up to three types of speakers for inclusion in the array. If speaker B and/or C are not to be included, select a **Type** of None on those tabs.

Click to show/hide the parameters.

Parameter	Description
Definition	
Type	Select a speaker from the default speaker data available in the library file; alternatively, select <Generic> to create a speaker with custom data and then save the speaker as a custom speaker in the library file. Select **Import** to import predefined speaker data only (such as width and weight) from a default content library file or other file containing speakers. The Import Speaker Type dialog box opens to select the file and type. A symbol is not imported from the file; only data is added
Brand	Specifies the speaker manufacturer
Model	Specifies the manufacturer's information
Save Type to Library	Saves the current speaker configuration as a new speaker type; enter the name of the speaker type. This custom speaker can then be selected from the **Type** list.
Delete Type from Library	Removes the speaker currently selected in **Type** from the library file; the current parameters remain as set, and the type becomes <Generic>
Attributes	
Front Width/Height	Indicates the width and height of the front face of the speaker
Back Width/Height	Indicates the width and height of the back face of the speaker; the back face cannot be larger than the front face
Depth	Specifies the speaker depth
Flip Orientation	Rotates the speaker 90° to place the speaker on its side
Tilt Reference	For speakers in an array, sets the reference for tilting the speaker to either the front or back face
Weight	Indicates the weight of the speaker
Self-Powered	Indicates whether the speaker uses an internal or remote amplifier
Dispersions	
Override Vertical Dispersion(s) with Array Angles	Calculates the dispersion based on the array's **Angle Relative to Preceding** value for each speaker set on the Array tab, rather than the individual speaker's vertical dispersion angles
Expanded Dispersion	When selected, allows you to enter range notes and dispersion area information for up to three ranges; deselect to enter only Overall information
Range 1/2/3	Enter notes for three dispersion ranges, such as a frequency range, key frequency, or crossover point. When **Expanded Dispersion** is deselected, the range is automatically set to Overall and only one set of notes can be entered
Horiz./Vert. Dispersion	Enter the horizontal and vertical dispersion angle(s), from 0 to 180°; ranges 2 and 3 are only available when **Expanded Dispersion** is selected
Throw Distance	Sets the throw distance reference for the dispersion lines; ranges 2 and 3 are only available when **Expanded Dispersion** is selected
Preview	Displays a preview of the speaker, which updates as parameters change; the arrow indicates the front face

6. Set up the array configuration on the Array tab.

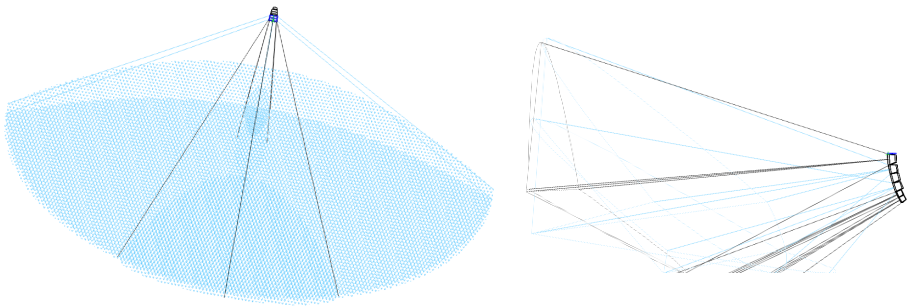
Click to show/hide the parameters.

Parameter	Description
Array Support	Specify how the speaker array is supported: Flown-Motors, Flown-Crank Lift, Ground-Stacked, or Tripod. Flown arrays place the speakers below the bumper. Ground-stacked arrays are inserted above the bumper. Arrays supported by a tripod do not include a bumper.
Bumper Type	Displays the bumper type that was selected on the Bumper tab (does not apply to tripod-supported arrays)
Bumper Angle	Sets the angle of the bumper, establishing a starting angle for the top mounting (does not apply to tripod-supported arrays)
First Speaker Shift	Controls the offset of the speaker stack from the bumper. Enter a positive value to shift the speaker stack towards the rear of the bumper; enter a negative value to offset the speaker stack towards the front of the bumper.
Tripod Height	For tripod-supported arrays, specifies the height at the top of the tripod
Definition	Lists the speakers that make up the array and their relative and actual dispersion angles, in order from top to bottom starting from below the bumper. Add speakers to the list by selecting one or more from the **Speaker Types to Add** list, and then clicking **Add/Swap Speaker(s).** To re-order the speakers, click the # column of the speaker to be moved, and drag it up or down the list. A horizontal line indicates where the speaker will be inserted in the current order.
Speaker Types to Add	Lists the speaker(s) defined on the Speaker tabs. These speakers are available for placement in the array.
Add/Swap Speaker(s)	Adds one or more selected speakers to the array; to swap speakers, select the speaker(s) to be replaced in the Definition list first. To add speakers without swapping, ensure that no speakers are highlighted in the Definition list.
Type Counts	Lists the number of each type of speaker, as well as the total number of speakers, included in the array
Delete Speaker(s)	Deletes the speaker(s) selected in the Definition list
Angle Relative to Preceding	Sets the angle of the speaker(s) selected in the Definition list. Individual speakers have range dispersion angle settings, but these can be overridden to use this angle set for the array instead. The **Rel°** and **Act°** angles in the Definition list update to reflect the current speaker angles.
Preview	Displays a preview of the speaker array, which updates as parameters change; the arrows indicate the front face

7. Click **OK** to create the speaker array. From the Object Info palette, click **Insert Speakers** to add the speakers to the drawing.

The speaker array properties can be edited in the Object Info palette. To edit the array, select the bumper. Individual speakers can be selected and certain parameters can be set independently, but since the speakers are controlled by the array, many of their parameters are not available (see “Inserting Speakers” on page 931 for parameter descriptions).

Click to show/hide the parameters.

Parameter	Description
Array Details	
Column ID	Specifies a name for the bumper-speaker combination that makes up the array
Location	Indicates the array location (such as Downstage Right)
Purpose	Indicates the purpose of the array (such as Center Fill)
Rack	Enter notes about the rack driving the speaker
Drive Line	Indicates information about the drive line feeding signal to the speaker
Notes	Enter miscellaneous notes such as rental source or stock number
Configure Array	Opens the Array Detail and Configuration dialog box, to view and edit speaker array parameters
Current Types	Lists the bumper and speakers that make up the array, along with speaker counts
Total Weight	Displays the total weight of all the array components, including the bumper; click **Insert Speakers** if not displayed
Bottom Box Trim	Displays the vertical trim of the bottom speaker; click **Insert Speakers** if not displayed
Listening Height	Sets the listening height reference for the speakers' dispersion lines. Listening height must be less than any speaker's elevation (Z value).
Override A/B/C Vert. Disp. with Array Angles	Calculates the dispersion based on the array's **Angle Relative to Preceding** value for each speaker set on the Array tab, rather than the individual speaker's vertical dispersion angles
Show Dispersion for Range 1/2/3	Draws the top, bottom, left, and right dispersion lines for each of the three dispersion ranges selected for display, for each speaker (or the Overall range if expanded dispersion is not selected). When applicable, a center reference line is drawn for each of the three dispersion ranges, for each of the speaker models in the array. An outline or hatch pattern representing the listening area coverage "shadow" is drawn when the throw distance and the vertical dispersion and tilt angles allow the speaker to cover the area at or below the **Listening Height**. 
Show Dispersion Hatch	Displays hatch fills in the coverage shadow of each enabled dispersion range
Class Speaker Array Parts	When first selected, automatically creates classes for the different parts of the speaker, for appearance and visibility control. This allows portions of the cabinets, grills, and other elements to be set to visible, grayed, or invisible. Different textures can be applied to the different part classes (Renderworks required), for a highly realistic appearance. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected speaker object. You may need to click **Update** to view any changes.

Parameter	Description
Parts Classes Prefix	When speaker parts are classed, enter a prefix for the class names so that they are sorted together
Insert Speakers	The first time the speaker array is created, click **Insert Speakers** after exiting the Array Detail and Configuration dialog box, to add the speakers to the array
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Speaker Object Labels” on page 939
Default Text Position	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters. For example, you may need to refresh the display to update the dispersion ranges and hatch after overriding the dispersion calculations.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Speakers and Speaker Arrays
Formatting Speaker Object Labels
Inserting Speakers

Formatting Speaker Object Labels

A variety of speaker and speaker array labels can be included on the drawing. Different options are available for speakers and speaker arrays. A speaker array, as well as the individual speakers that make up the array, can be labeled.

To select labels for display and format the text:

1. Select a speaker or speaker array object (bumper). From the Object Info palette, click **Text Options**.

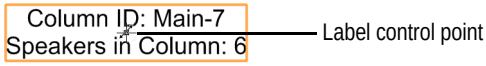
   The Text Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Optional Elements to Show	Select items to add to the text label; different label elements are available for speakers and speaker arrays
Show Element Labels	Includes the name of the element along with the value; deselect this option to display the parameter value only
Text Attributes	
Keep Text Horizontal	Maintains the text in a horizontal position even when the object is rotated; deselect this option to rotate the label along with the object
Fill Text Background	Allows a text background fill to be used for all speaker object text; by default, white is used as the fill color. To specify a different fill color, select the Object Info palette option to class the parts of the speaker object, and then select a fill color for the class of the speaker text. The class should be set to **Use at Creation**.
Text formatting options	Specify the text size, color, and alignment

2. Specify the text label elements and formatting, and then click **OK**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

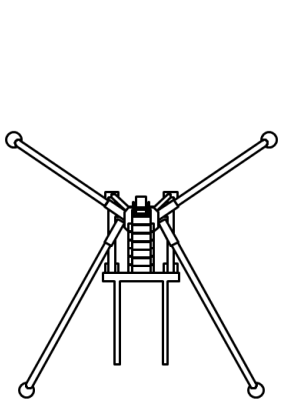
Once the label has been added to the drawing, it can be moved by clicking and dragging the label control point. Click **Default Text Position** from the Object Info palette of a selected speaker or array object to restore the text label(s) to the original location.



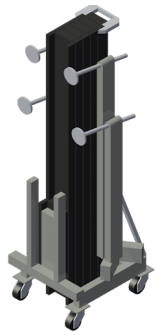
Inserting Speakers and Speaker Arrays
Inserting Speakers
Inserting Speaker Arrays

Inserting a Stage Lift

A stage lift object can be added to a drawing to represent the typical adjustable support device commonly found on stage for supporting speakers, trusses, and other stage equipment. The stage lift can be shown extended or collapsed, with a specific appearance and fork position, and can be labeled.


2D view


3D view


Stored 3D view

To insert a stage lift:

1. Click the **Stage Lift** tool from the Spotlight tool set.
2. Click once in the drawing to set the object's position. Click again to set the object's rotation.
3. If this is the first time a stage lift is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all stage lifts placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

The stage lift parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Stored	When selected, folds the stage lift and fully retracts the mast; forks and support legs are placed in the stored position. Certain parameters are not available when the stage lift is in stored position.
Fork Position	Select whether the forks are in the up or down position, or whether the lift has no forks

Parameter	Description
Height	Sets the height of the forks; the mast extends automatically to add height to the stage lift, up to the maximum possible height
Maximum Height	Displays the maximum possible height that can be set for the forks, when all mast sections have been extended
Width	Specifies the spacing of the forks, as measured inside the forks; set the width to 0 to hide the forks, but keep the crossbar visible
Orientation	Select whether the forks are oriented horizontally or vertically
Forks	Select gray or black for the color of the forks
Mast	Select black or silver for the color of the mast
3D Colors	
Legs and Base	Select gray or black for the color of the legs and base when the lift is viewed in 3D
Hardware	Select silver or black for the color of the hardware when the lift is viewed in 3D
Name, Lift Type, Location, Load Type, Load Weight, Note	Adds information which can be placed on the drawing with the **Text Options**
Class Stage Lift Parts	When first selected, automatically creates classes for the different parts of the stage lift for appearance and visibility control. This allows portions of the stage lift to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected stage lift object.
Parts Classes Prefix	When stage lift parts are classed, creates a prefix for the class names so that they are sorted together
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Stage Lift Labels” on page 941
Default Text Position	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Formatting Stage Lift Labels

Formatting Stage Lift Labels

A variety of stage lift labels can be included on the drawing. Text must be entered into the associated field of the Object Info palette for the label to be included.

To select labels for display and format the text:

1. Select a stage lift object. From the Object Info palette, click **Text Options**.

   The Text Options dialog box opens.

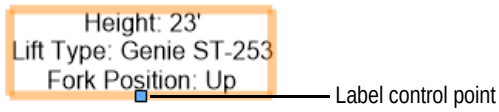
Click to show/hide the parameters.

Parameter	Description
Elements to Show	Select items to add to the text label
Text Attributes	
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Keep Text Horizontal	Maintains the text in a horizontal position even when the object is rotated; deselect this option to rotate the label along with the object
Fill Text Background	Allows a text background fill to be used for all stage lift text; by default, white is used as the fill color. To specify a different fill color, select the Object Info palette option to class the parts of the stage lift object, and then select a fill color for the class of the stage lift text. The class should be set to **Use at Creation**.
Text formatting options	Specify the text size, color, and alignment

2. Specify the text label elements and formatting, and then click **OK**.

   Once the label has been added to the drawing, it can be moved by clicking and dragging the label control point. Click **Default Text Position** from the Object Info palette of a selected stage lift object to restore the text label(s) to the original location.



Inserting a Stage Lift

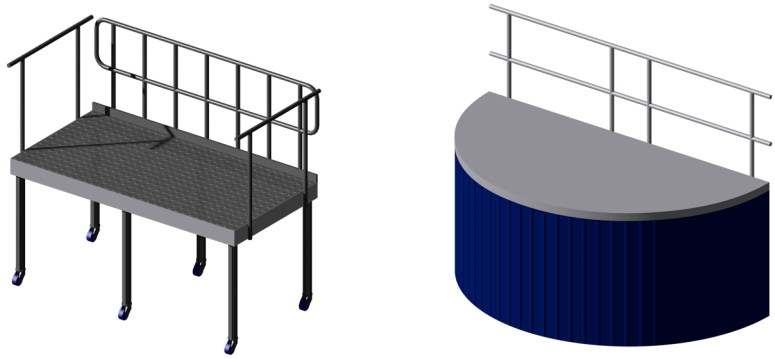
Inserting Stage Structures

Vectorworks includes several different types of rectangular, round, and free-form stage structures, along with the specialized, industry-standard steps and ramps to access them. These theatrical and event staging structures can range from simple, basic forms to complex structures with customized textures for a realistic appearance (Renderworks required). The event design commands also make use of these structures.

Inserting a Stage Deck
Inserting a Stage Plug
Inserting Stage Steps
Inserting a Stage Ramp

Inserting a Stage Deck

The **Stage Deck** tool inserts rectangular and rounded portable stage structures into a drawing. The type, shape, size, appearance, and presence of railings can be customized. Place several stage decks to create a large stage area.

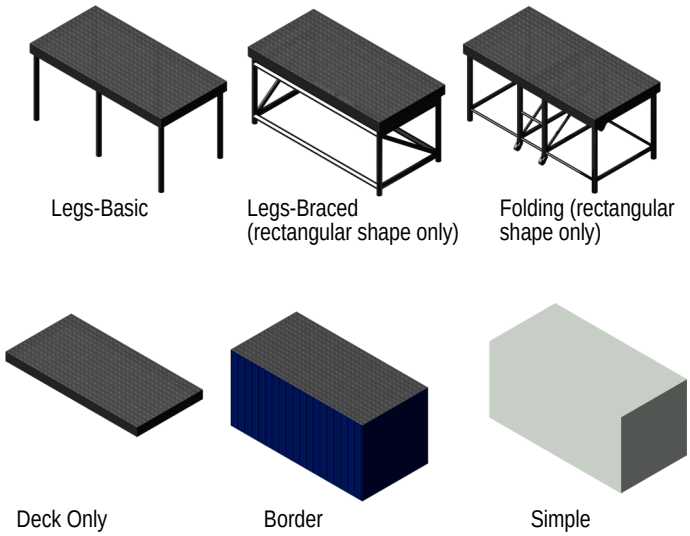


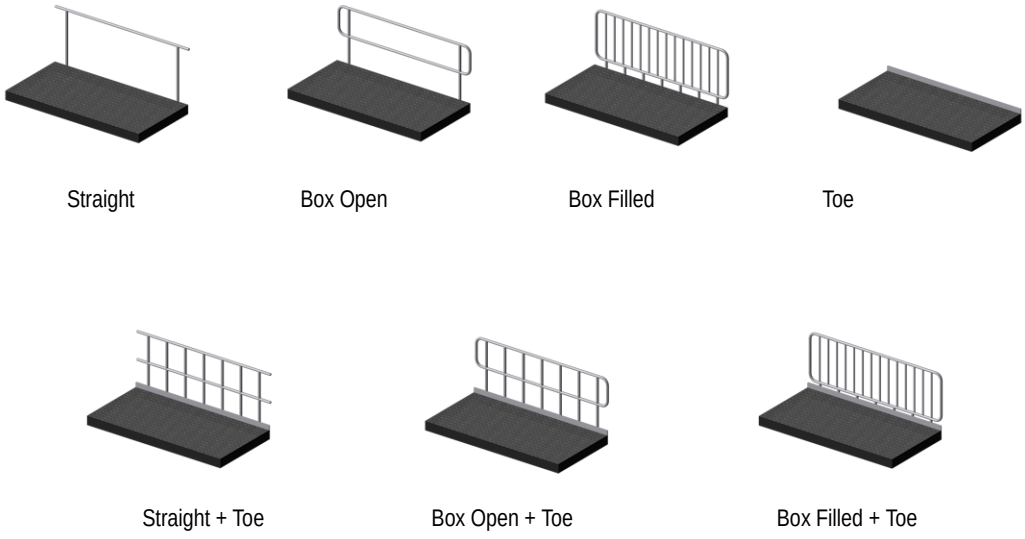
 To insert a stage deck:

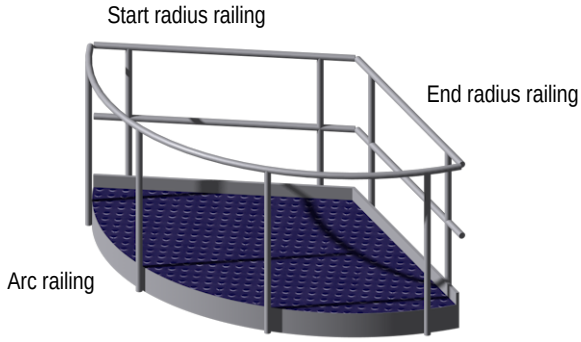
1. Click the **Stage Deck** tool from the Spotlight tool set.
2. Click once in the drawing to set the object's position. Click again to set the object's rotation.
3. If this is the first time a stage deck is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all stage decks placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

The stage deck parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Deck Shape	Creates a rectangular or round stage
Height	Specifies the overall stage height
Depth/Width (rectangular shape)	Specifies the dimensions of a rectangular stage deck
Radius/Diameter/ Sweep Angle (round shape)	Specifies the size of a round stage deck
Top Surface Area	Displays the calculated surface area of the stage
Structure	Select the type of stage construction, from simple to complex 
Deck Details	
Hide Deck	For the Legs-Braced structure types, hides the deck and shows the braced leg supports only
Top Thickness	Sets the thickness of the deck. The thickness of the deck cannot be set for Border and Simple stage deck structures.
Top Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Trim Height	Sets the height of the trim around the stage

Parameter	Description
Trim Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Border Color/Texture	For the Border structure type, displays the texture (Renderworks required) or color selected in the 3D Options
Leg/Structure Details	
Diameter	For structure types with legs, sets the diameter of the leg supports
Profile	For structure types with legs, select a round, square, or octagonal shape for the legs
Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Reference Spacing	For Legs-Basic and Legs-Braced, sets an approximate spacing value for the legs; the exact spacing is determined automatically by evenly dividing the length of the deck
Total Legs	For structure types with legs, displays the number of legs
Add Casters	For the Legs-Basic and Legs-Braced (rectangular shape only) structure types, adds casters to the legs to create a rolling platform
Wheel Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Rail Details	All railing bars are automatically evenly spaced
Front/Back/Left/ Right (rectangular shape)	Select the type of railing to add to each side of the stage deck, or select None for no railing  Straight Box Open Box Filled Toe Straight + Toe Box Open + Toe Box Filled + Toe

Parameter	Description
Start Radius/ End Radius/ Arc (round shape)	For round deck shapes, select the type of railing to add to each side of the stage deck, or select None for no railing. Rounds decks are limited to Straight, Toe, or Straight + Toe railing types. 
Height	Sets the height of the top of the railing
Add'l Horiz. Bars (Straight and Straight + Toe)	Specifies the number of additional horizontal bars to add to the railing
Add'l F/B Uprights (rectangular shape)	For rectangular deck shapes, adds additional vertical bars to the front and back rails
Add'l L/R Uprights (rectangular shape)	For rectangular deck shapes, adds additional vertical bars to the left and right rails
Add'l Radii Uprights (round shape)	For round deck shapes, adds additional vertical bars to the start and end radius rails
Add'l Arc Uprights (round shape)	For round deck shapes, adds additional vertical bars to the arc rails
Profile	Select a round, square, or octagonal shape for the railings
Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Class Stage Deck Parts	When first selected, automatically creates classes for the different parts of the stage deck, for appearance and visibility control. This allows portions of the stage deck to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected stage deck object.
Parts Classes Prefix	When stage deck parts are classed, creates a prefix for the class names so that they are sorted together
Purpose/ Set Cart ID	Enter information about the stage which can be placed on the drawing with **Text Options** (does not affect the stage appearance)
Note	Adds a note, which can be placed on the drawing with the **Text Options**
3D Options	Specifies the appearance of applicable deck elements in 3D views. The 3D Options dialog box opens. For each available portion of the stage, either select the color from the **Color** list, or, when Renderworks is installed, select Custom Texture and then select the texture from the **Texture** list. The Object Info palette displays the color or texture selections in the relevant section. The appearance of Simple stage decks is set from the Attributes palette.

Parameter	Description
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Stage Labels” on page 948
Default Text Positions	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

Formatting Stage Labels
Inserting a Stage Plug

Inserting a Stage Plug

The **Stage Plug** tool inserts custom shaped stage structures into a drawing. The type, shape, size, and appearance can be customized.

To insert a stage plug:

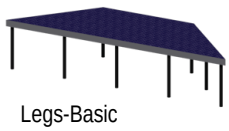
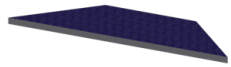
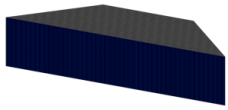
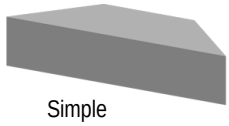
1. Click the **Stage Plug** tool from the Spotlight tool set.
2. Click on the appropriate mode in the Tool bar to select the boundary creation method of the stage.

   For more information on the **Polyline** tool modes, see “Creating Polylines” on page 298.
3. Click to set the stage’s start point.
4. Click to set the end of the segment and the beginning of the next. Continue drawing segments in this manner until the stage object is complete.
5. If this is the first time a stage plug is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all stage plugs placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

The stage plug can be reshaped with the **Reshape** tool; see “Reshaping Objects” on page 1043. The stage plug parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Stage Height	Specifies the overall stage height
Top Surface Area	Displays the calculated surface area of the stage

Parameter	Description
Structure	Select the type of stage construction  Legs-Basic  Deck Only  Border  Simple
Deck Details	
Top Thickness	Sets the thickness of the deck. The thickness of the deck cannot be set for Border and Simple stage plug structures.
Top Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Trim Height	Sets the height of the trim around the stage
Trim Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Leg Details	
Diameter	For the Legs-Basic structure type, sets the size of the leg supports
Profile	For the Legs-Basic structure type, select a round, square, or octagonal shape
Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Reference Spacing	For the Legs-Basic structure type, sets an approximate spacing value for the legs; the exact spacing is determined automatically by evenly dividing the length of the deck
Total Legs	For the Legs-Basic structure type, displays the number of legs
Add Casters	For the Legs-Basic structure type, adds casters to the legs to create a rolling platform
Wheel Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Class Stage Plug Parts	When first selected, automatically creates classes for the different parts of the stage plug, for appearance and visibility control. This allows portions of the stage plug to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected stage plug object.
Parts Classes Prefix	When stage plug parts are classed, creates a prefix for the class names so that they are sorted together
Purpose/ Set Cart ID	Enter information about the stage which can be placed on the drawing with **Text Options** (does not affect the stage appearance)
Note	Adds a note, which can be placed on the drawing with the **Text Options**

Parameter	Description
3D Options	Specifies the appearance of applicable deck elements in 3D views. The 3D Options dialog box opens. For each available portion of the stage, either select the color from the **Color** list, or, when Renderworks is installed, select Custom Texture and then select the texture from the **Texture** list. The Object Info palette displays the color or texture selections in the relevant section. The appearance of Simple stage plugs is set from the Attributes palette.
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see “Formatting Stage Labels” on page 948
Default Text Positions	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

Formatting Stage Labels
Inserting a Stage Deck

Formatting Stage Labels

A variety of stage labels can be included on the drawing. For some options, text must be entered into the associated field of the Object Info palette for the label to be included.

To select labels for display and format the text:

1. Select a stage deck or stage plug object. From the Object Info palette, click **Text Options**.

   The Text Options dialog box opens.

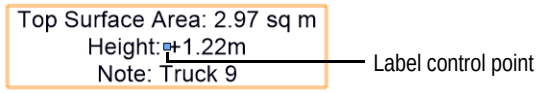
Click to show/hide the parameters.

Parameter	Description
Elements to Show	
Available options	Lists available labels that can be included; the distance units for some of the labels can be displayed with both primary and secondary units. Select whether to use the default drawing units or choose a different unit from the list.
Include “+” with Height	For labels with height values, adds a “+” prefix to the measurement
Show Element Labels	Includes the name of the element along with the value; deselect this option to display the parameter value only
Text Attributes	
Keep Text Horizontal	Maintains the text in a horizontal position even when the object is rotated; deselect this option to rotate the label along with the object
Fill Text Background	Allows a text background fill to be used for all stage object text; by default, white is used as the fill color. To specify a different fill color, select the Object Info palette option to class the parts of the stage object, and then select a fill color for the class of the stage text. The class should be set to **Use at Creation**.

Parameter	Description
Text formatting options	Specify the text size, color, and alignment

2. Specify the text label elements and formatting, and then click **OK**.

Once the label has been added to the drawing, it can be moved by clicking and dragging the label control point. Click **Default Text Positions** from the Object Info palette of a selected stage object to restore the text label(s) to the original location.

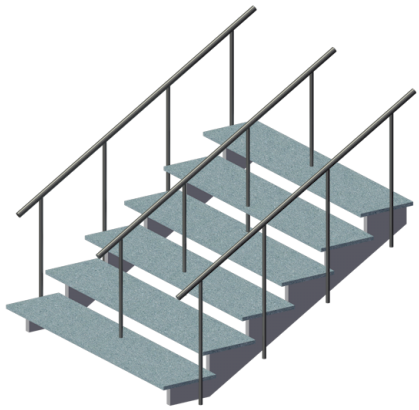


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Stage Deck
Inserting a Stage Plug

Inserting Stage Steps

The **Stage Steps** tool inserts portable steps typically used with temporary stage structures into a drawing. The type, shape, size, appearance, and presence of railings can be customized.

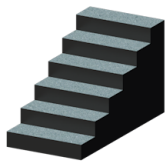
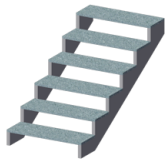
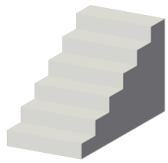


 To insert stage steps:

1. Click the **Stage Steps** tool from the Spotlight tool set.
2. Click once in the drawing to set the object's position. Click again to set the object's rotation.
3. If this is the first time a set of stage steps is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all stage steps placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

The stage step parameters can be edited in the Object Info palette.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Structure	Select the style of stage steps; self-adjusting steps have a fixed stringer length and number of steps, and the step rise automatically adjusts to fit the stage height. For the other types of steps, the step parameters are determined according to the selected **Create Based On** option.  Self-Adjusting  Frame Free-Standing  Solid Free-Standing  Open Free-Standing  Simple
Create Based On	When steps are not self-adjusting, specifies the method used to determine the step parameters, either by setting stage height and number of steps, or by setting the number of steps and the step rise. Depending on the selection, certain step parameters become available, while the other parameters are calculated for display. • **Stage Height and Number of Steps**: Calculates the height of the steps and the rise automatically based on the height of the stage and the specified number of steps • **Number of Steps and Step Rise**: Calculates the height of the steps and the stage height based on the specified number of steps and the step rise
Stage Height	Sets or displays the height of the stage
Step Unit Width	Specifies the width of each step
Stringer Length (Self-adjusting steps)	Specifies the length of the stringers for self-adjusting steps
Stringer Width (self-adjusting and open free-standing steps)	Sets the width of the stringers for steps with stringers
Stringer Type (open free-standing steps)	Select whether the stringers are located inside and under the treads, or outside the treads (the stringer width setting affects the available step area)
Add'l Stringers (open free-standing steps)	Adds one or more additional support stringers to the steps, spaced evenly based on step width
Side Stringer Indent (open free-standing steps, inside stringers only)	Moves the outer stringers in (underneath the steps) on each side of the stair by the specified distance
Step Details	

Parameter	Description
Number of Steps	Specifies the number of steps
Add Top Step at Stage Height	Adds an additional step at the top of the flight, set to the same height as the stage
Top Step Height	Displays the height of the top step
Single Step Rise	Sets the distance from one step to the next
Step Thickness	Specifies the thickness of each step
Single Step Depth	Specifies the depth of each step
Nosing Depth (open free-standing steps)	Sets the nosing distance (the depth the step protrudes from the stringer)
Top Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Trim Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Pitch (Calculated)	
Angle	Displays the pitch of the stair as an angle
Ratio (rise:run)	Displays the pitch of the stair as a rise-over-run ratio
Percent (Grade)	Displays the pitch of the stair as a grade percentage
Leg/Structure Details	
Profile (frame free-standing steps)	For the Legs-Basic structure type, select a round, square, or octagonal shape
Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Rail Details	
Show Left/ Right Rail	Select whether to show a rail on the left only, right only, or both sides. Rail parameters apply to both sides.
Height	Specifies the height of the top rail
Add'l Whole Rails (simple, solid free-standing, or open free-standing steps)	Adds one or more additional rail sets to the steps, spaced evenly based on step width; this parameter is only available when both left and right rails are selected for display
Add'l Horiz Bars	Adds additional horizontal bars to the rails; not available for self-adjusting steps
Add'l Uprights	Adds additional upright bars to the rails; not available for self-adjusting steps
Profile	Select a round, square, or octagonal rail shape
Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Draw Arrow	Select whether to display an arrow in Top/Plan view; the arrow can point up or down. In the **Text Options**, select the arrow text label to display the word "Up" or "Down" on the drawing.
Arrow Size	When drawing an arrow, sets the arrow head size in drawing units
Class Stage Steps Parts	When first selected, automatically creates classes for the different parts of the stage steps, for appearance and visibility control. This allows portions of the steps to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected stage steps object.

Parameter	Description
Parts Classes Prefix	When stage steps parts are classed, creates a prefix for the class names so that they are sorted together
Set Cart ID	Enter information about the steps which can be placed on the drawing with **Text Options** (does not affect the steps' appearance)
Note	Adds a note, which can be placed on the drawing with the **Text Options**
3D Options	Specifies the appearance of applicable step elements in 3D views. The 3D Options dialog box opens. For each available portion of the steps, either select the color from the **Color** list, or, when Renderworks is installed, select Custom Texture and then select the texture from the **Texture** list. The Object Info palette displays the color or texture selections in the relevant section. The appearance of Simple stage steps is set from the Attributes palette.
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see "Formatting Stage Step Labels" on page 952
Default Text Positions	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Formatting Stage Step Labels

Formatting Stage Step Labels

A variety of stage step labels can be included on the drawing. For some options, text must be entered into the associated field of the Object Info palette for the label to be included.

To select labels for display and format the text:

1. Select a stage steps object. From the Object Info palette, click **Text Options**.

   The Text Options dialog box opens.

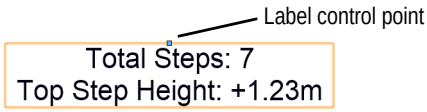
Click to show/hide the parameters.

Parameter	Description
Elements to Show	
Available options	Lists available labels that can be included; the distance units for some of the labels can be displayed with both primary and secondary units. Select whether to use the default drawing units or choose a different unit from the list.
Arrow Label	When an up or down arrow is selected for **Draw Arrow** in the Object Info palette, adds an "Up" or "Down" label to the drawing in Top/Plan view
Include "+" with Height	For labels with height values, adds a "+" prefix to the measurement
Show Element Labels	Includes the name of the element along with the value; deselect this option to display the parameter value only
Text Attributes	
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Keep Text Horizontal	Maintains the text in a horizontal position even when the object is rotated; deselect this option to rotate the label along with the object
Fill Text Background	Allows a text background fill to be used for all stage steps object text; by default, white is used as the fill color. To specify a different fill color, select the Object Info palette option to class the parts of the stage steps object, and then select a fill color for the class of the steps' text. The class should be set to **Use at Creation**.
Text formatting options	Specify the text size, color, and alignment

2. Specify the text label elements and formatting, and then click **OK**.

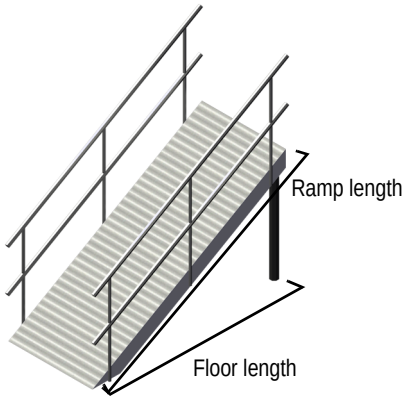
Once the label has been added to the drawing, it can be moved by clicking and dragging the label control point. Click **Default Text Position** from the Object Info palette of a selected stage steps object to restore the text label(s) to the original location.



Inserting Stage Steps

Inserting a Stage Ramp

The **Stage Ramp** tool inserts the type of portable ramp typically used with temporary stage structures into a drawing. The type, shape, size, appearance, and presence of railings can be customized.

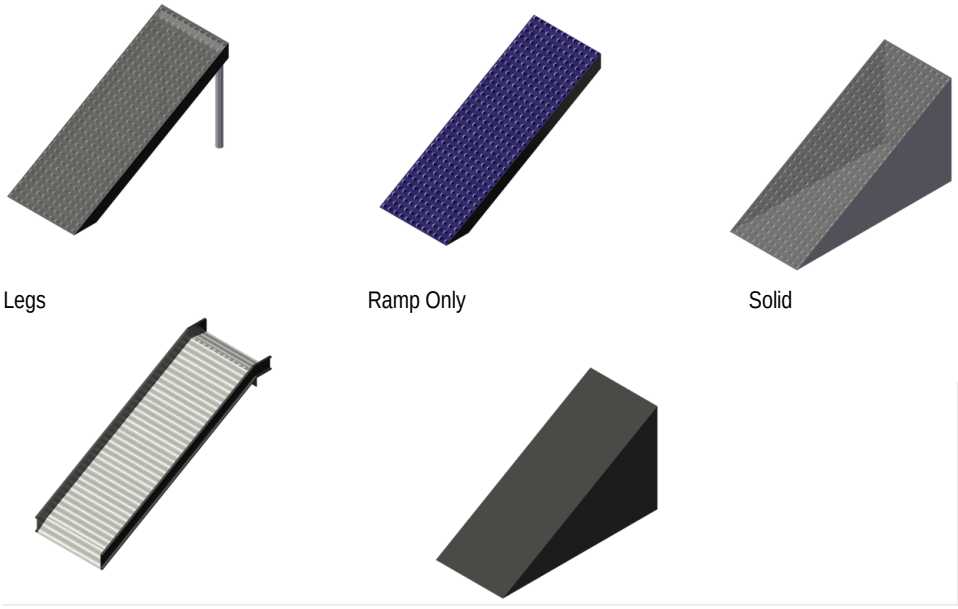


To insert a stage ramp:

1. Click the **Stage Ramp** tool from the Spotlight tool set.
2. Click once in the drawing to set the object's position. Click again to set the object's rotation.
3. If this is the first time a stage ramp is placed on the drawing, the object properties dialog box opens. Specify the default preferences, which apply to all stage ramps placed subsequently in this drawing. Properties can be edited later in the Object Info palette. Click **OK**.

The stage ramp parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Structure	Select the style of stage ramp 
Height	Sets the height of the top of the ramp
Width	Specifies the width of the ramp, not including any rails
Create Based On	Specifies the method used to determine the ramp parameters, by setting ramp length, floor length, or pitch. Depending on the selection, certain ramp parameters become available, while the other parameters are calculated for display. • **Ramp Length**: Calculates the floor length and the pitch automatically based on the specified ramp length • **Floor Length**: Calculates the ramp length and the pitch automatically based on the specified floor length • **Pitch Angle**: Calculates the ramp length and the floor length automatically based on the specified pitch angle • **Pitch Ratio**: Calculates the ramp length and the floor length automatically based on the specified pitch ratio • **Pitch Percent**: Calculates the ramp length and the floor length automatically based on the specified pitch percentage
Ramp Length	Sets or displays the length of the ramp
Floor Length	Sets or displays the length from the bottom of the ramp to the top of the ramp, as projected to the floor
Pitch	
Angle	Sets or displays the pitch of the ramp as an angle
Ratio (rise:run)	Sets or displays the pitch of the ramp as a rise-over-run ratio
Percent (Grade)	Sets or displays the pitch of the ramp as a grade percentage

Parameter	Description
Ramp Deck Details	
Top Thickness	Specifies the thickness of the ramp
Top Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Trim Height	Sets the thickness of the ramp trim
Trim Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Leg/Structure Details (ramp with legs only)	
Diameter	Sets the diameter of the ramp legs
Profile	Select a round, square, or octagonal shape for the legs
Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Reference Spacing	Sets an approximate spacing value for the legs; the exact spacing is determined automatically by evenly dividing the length of the ramp
Total Legs	Displays the number of legs
Rail Details	
Show Left/ Right Rail	Select whether to show a rail on the left only, right only, or both sides. Rail parameters apply to both sides.
Height	Specifies the height of the top rail
Add'l Whole Rails	Adds one or more additional rail sets to the ramp, spaced evenly based on ramp width; this parameter is only available when both left and right rails are selected for display
Add'l Horiz Bars	Adds additional horizontal bars to the rails
Add'l Uprights	Adds additional upright bars to the rails
Profile	Select a round, square, or octagonal rail shape
Color/Texture	Displays the texture (Renderworks required) or color selected in the 3D Options
Draw Arrow	Select whether to display an arrow in Top/Plan view; the arrow can point up or down. In the **Text Options**, select the arrow text label to display the word "Up" or "Down" on the drawing.
Arrow Size	When drawing an arrow, sets the arrow head size in drawing units
Class Stage Ramp Parts	When first selected, automatically creates classes for the different parts of the stage ramp, for appearance and visibility control. This allows portions of the ramp to be set to visible, grayed, or invisible. Once the classes have been created, this check box toggles whether class visibility changes apply to the selected stage ramp object.
Parts Classes Prefix	When stage ramp parts are classed, creates a prefix for the class names so that they are sorted together
Set Cart ID	Enter information about the ramp which can be placed on the drawing with **Text Options** (does not affect the stage appearance)
Note	Adds a note, which can be placed on the drawing with the **Text Options**

Parameter	Description
3D Options	Specifies the appearance of applicable ramp elements in 3D views. The 3D Options dialog box opens. For each available portion of the ramp, either select the color from the **Color** list, or, when Renderworks is installed, select Custom Texture and then select the texture from the **Texture** list. The Object Info palette displays the color or texture selections in the relevant section. The appearance of Simple stage steps is set from the Attributes palette.
Text Options	Opens the Text Options dialog box, to enable the display and format the text of labels; see "Formatting Stage Step Labels" on page 952
Default Text Positions	Restores text labels to their default positions
Update	Updates the object when changes have been made to the Object Info palette parameters

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Formatting Stage Ramp Labels

Formatting Stage Ramp Labels

A variety of stage ramp labels can be included on the drawing. For some options, text must be entered into the associated field of the Object Info palette for the label to be included.

To select labels for display and format the text:

1. Select a stage ramp object. From the Object Info palette, click **Text Options**.

   The Text Options dialog box opens.

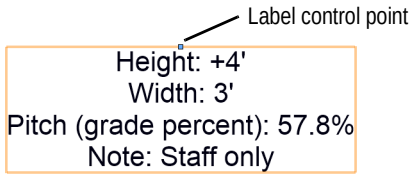
Click to show/hide the parameters.

Parameter	Description
Elements to Show	
Available options	Lists available labels that can be included; the distance units for some of the labels can be displayed with both primary and secondary units. Select whether to use the default drawing units or choose a different unit from the list.
Arrow Label	When an up or down arrow is selected for **Draw Arrow** in the Object Info palette, adds an "Up" or "Down" label to the drawing in Top/Plan view
Include "+" with Height	For labels with height values, adds a "+" prefix to the measurement
Show Element Labels	Includes the name of the element along with the value; deselect this option to display the parameter value only
Text Attributes	
Keep Text Horizontal	Maintains the text in a horizontal position even when the object is rotated; deselect this option to rotate the label along with the object
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Fill Text Background	Allows a text background fill to be used for all stage ramp text; by default, white is used as the fill color. To specify a different fill color, select the Object Info palette option to class the parts of the ramp object, and then select a fill color for the class of the ramp text. The class should be set to **Use at Creation**.
Text formatting options	Specify the text size, color, and alignment

2. Specify the text label elements and formatting, and then click **OK**.

Once the label has been added to the drawing, it can be moved by clicking and dragging the label control point. Click **Default Text Positions** from the Object Info palette of a selected stage ramp object to restore the text label(s) to the original location.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Stage Ramp
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

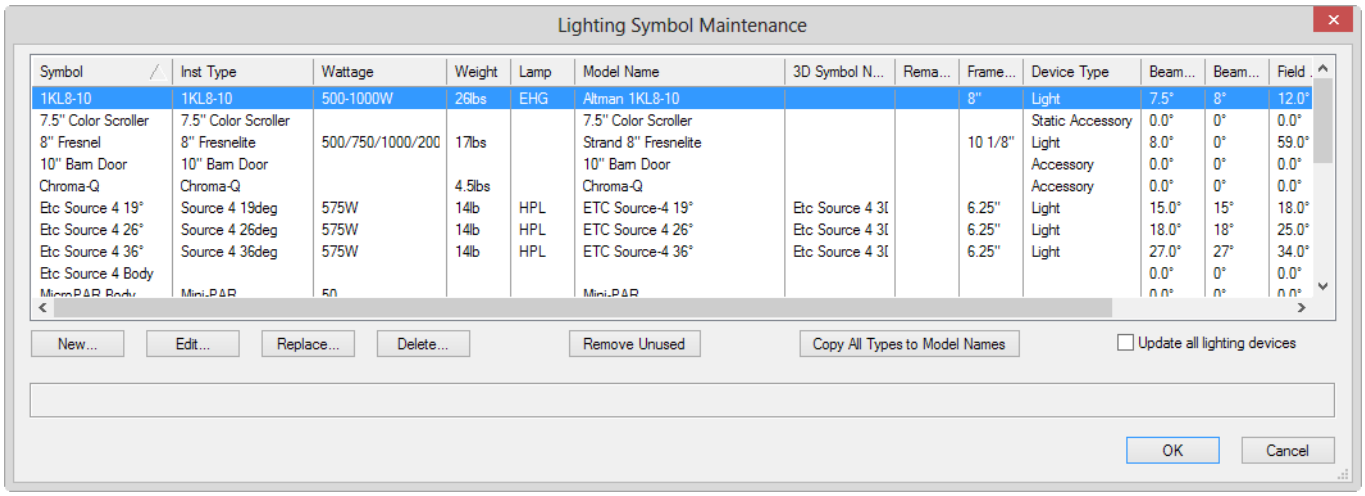
Lighting Symbol Maintenance

Manage, edit, and update the symbols and data for all the lighting symbols in the file from a single location. Easily spot inconsistencies and missing data, and fix the issues for all existing symbols or just for future symbol placement.

To maintain the data and symbols for all the lighting symbols in the file:

1. Select **Spotlight > Reports > Lighting Symbol Maintenance**.

   The Lighting Symbol Maintenance dialog box opens.



Click to show/hide the parameters.

Parameter	Description
Lighting symbol list	Lists all symbols in the file with a Light Info (or Light Info M, for metric measurements) record attached. The columns display the data contained in the attached Light Info record. Sort the symbols in ascending or descending alphabetical order by clicking in the first column header.
New	Opens the Choose a Symbol dialog box, listing all the symbol definitions in the file without the Light Info (or Light Info M) record attached. Select a symbol to add the Light Info record to the symbol definition and add the symbol to the maintenance list, where its data can then be edited.
Edit	Opens the Edit Instrument Data dialog box, displaying the record data for the selected row(s) (alternatively, double-click on the row to edit).
Replace	Opens the Choose a Symbol dialog box, listing all the symbol definitions in the file that are not in the maintenance list. Select a symbol to replace the symbol from the currently selected row of the maintenance list. The existing data from the selected row remain unchanged; only the symbol is replaced.
Delete	Deletes the Light Info (or Light Info M) record from the symbol in the currently selected row of the maintenance list, and removes the row from the list. The symbol is not removed from the file, but the symbol no longer appears in reports about the lighting device.
Remove Unused	Removes the Light Info (or Light Info M) record from all symbols in the maintenance list which are currently not used in a lighting device. The items are removed from the maintenance list, but the symbols are not removed from the file.

Parameter	Description
Copy All Types to Model Names	For uniformity, copies the data from the **Inst Type** column to the **Model Name** column. This eliminates inconsistencies and missing model name data.
Update all lighting devices	When selected, the symbol definition edits that have been made affect all existing and future symbol instances in the file, potentially changing existing lighting devices in the drawing. Deselect the option to only affect future symbol placement.

2. When finished with lighting symbol maintenance tasks, click **OK**.
3. If any lighting instrument data has changed, but **Update all lighting devices** was not selected, an alert message displays.

   Click **Yes** to update all existing and future symbol instances in the file. Select **No** to update only future symbol instances.

Lighting Inventory Setup

The current status of available lighting device inventory is specified with the **Lighting Inventory Setup** command. The number of available devices can be included in instrument summaries.

To establish and maintain inventory counts:

1. Select **Spotlight > Reports > Lighting Inventory Setup**.

   The Lighting Inventory Setup dialog box opens.
2. Select each device type row, and enter the number of available instruments in the **Quantity**.

   The devices can be sorted by clicking in the **Type** column header.
3. Click **OK**.

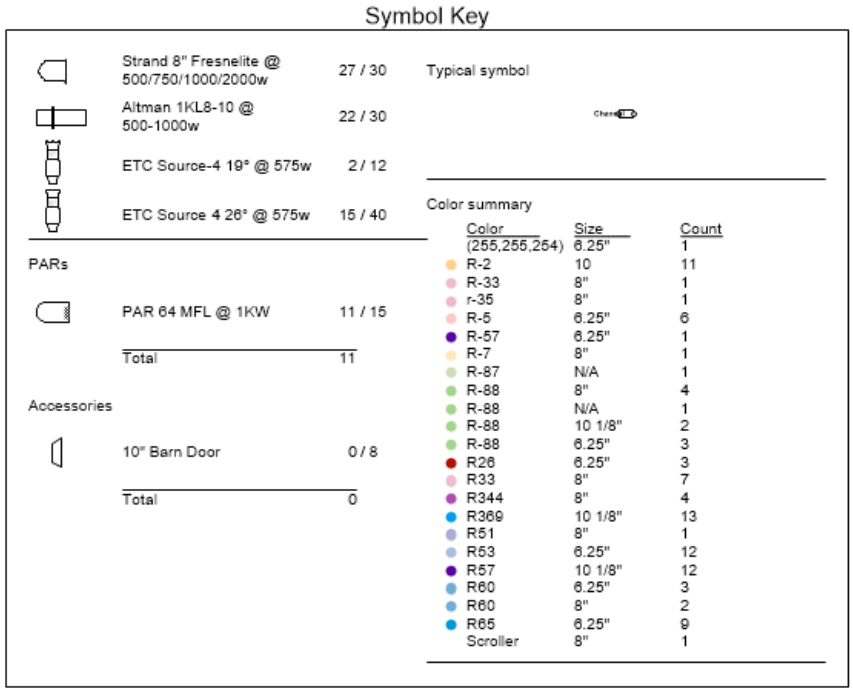
   The inventory report can be included as a report when generating paperwork.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Instrument Summaries
Generating Paperwork

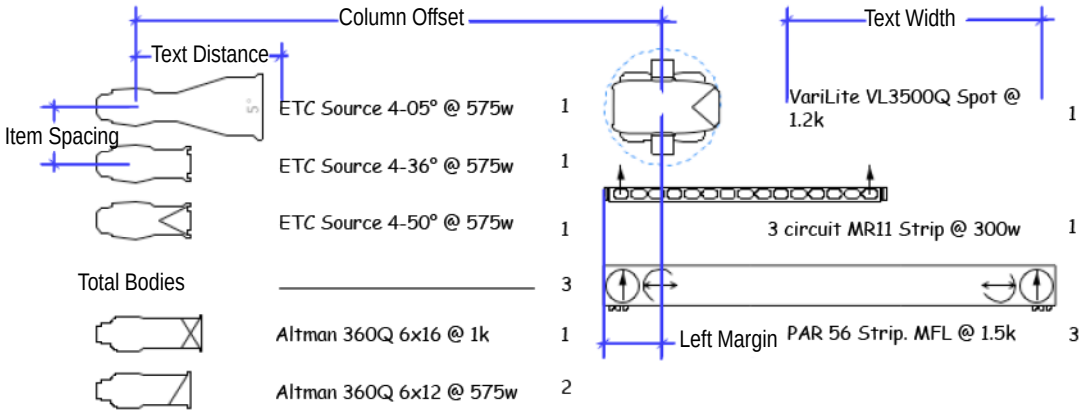
Creating Instrument Summaries

Statistics on the instruments and accessories used in the light plot help with show organization. Instrument summaries can be created for the all instruments (see “Creating an Instrument Summary” on page 961), and/or for specific lighting positions (see “Creating an Instrument Summary for a Lighting Position” on page 966). The summary, which can be custom formatted, can show the symbol thumbnail and name, the number of each kind of instrument and accessories in use, display total counts, and add special elements like notes, column breaks, circuit summaries, typical label legend layout, and other items. Counting can be subtotaled to count instrument bodies and lenses. Symbols which have the same instrument type are counted together, allowing for alternate versions of a single type. The summary can compare its information to the current inventory so that the designer can determine how many instruments remain in the inventory, or whether more instruments have been specified than are currently present in the inventory.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Creating an Instrument Summary
Creating an Instrument Summary for a Lighting Position

Creating an Instrument Summary



To create an instrument summary:

1. Click the **Instrument Summary** tool from the Spotlight tool set.
2. Click **Preferences** from the Tool bar.

The Instrument Summary Settings dialog box opens. Default parameter settings made here, both for global instrument summaries and lighting position summaries, apply to instrument summaries placed later in the file. These default sets are saved separately and are in effect depending on how the summary is inserted. Taking the time to establish the defaults makes it much easier to insert summaries for lighting positions or for the entire drawing with the desired defaults already specified.

Click to show/hide the parameters.

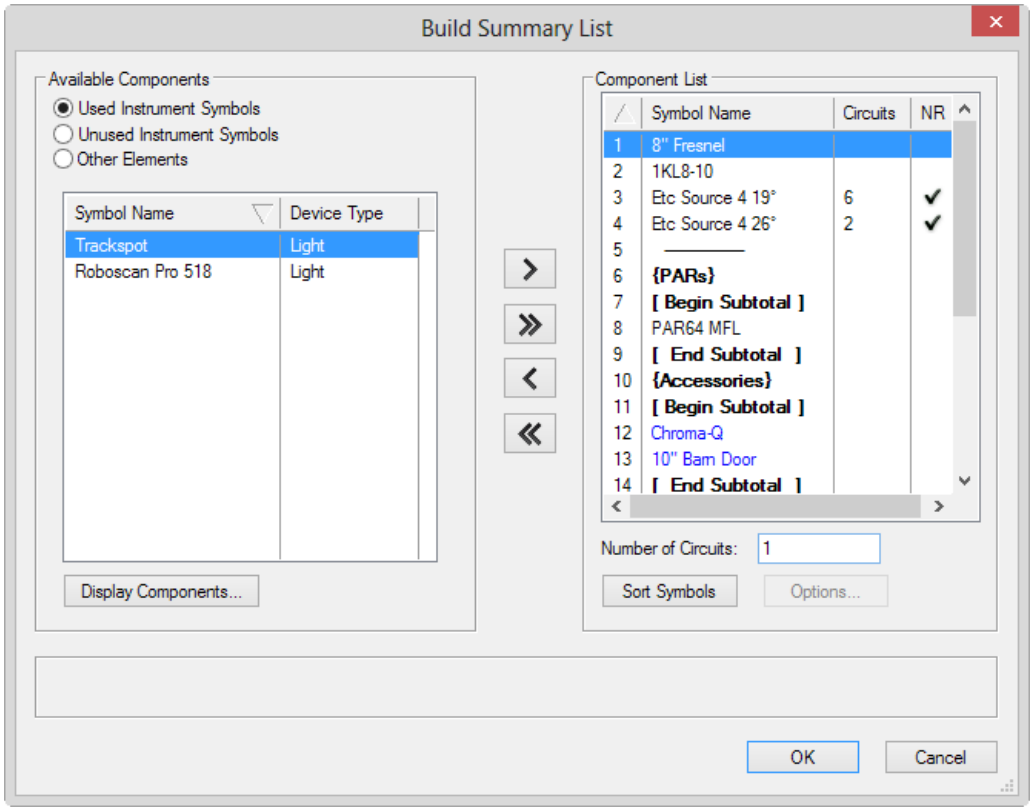
Parameter	Description
Summary Defaults for	Select whether to specify the default settings for the summary when inserting it with the **Instrument Summary** tool (select Instruments) or from a lighting position (select Positions). This option is not available from the Object Info palette after the summary has been placed.
Build List	Opens the Build Summary List dialog box, for specifying the components to include in the summary
Width	Sets the total width of the summary
Column Offset	Sets the distance between columns
Instruments	
Hide Unused Types	When selected, excludes instrument symbols which are not currently used in the drawing (these instruments use a blue font in the Build Summary List dialog box), and only displays symbols which are used by a lighting device
Show Symbols	Includes a thumbnail graphic of each lighting instrument symbol in the key
Auto Rotate	Orients symbols automatically to minimize the amount of space required
Scale	Scales symbols by the specified proportion; this allows the instrument summary to be placed on a sheet layer without displaying the symbols at 1:1 scale
Show Wattages	Appends "@ [wattage]" to the instrument type. A blank wattage or a wattage of 0 does not display.
Show Weights	Displays the unit's weight, as set in the default parameters of the symbol (in the Light Info Record, or for metric drawings, Light Info Record M)
Show Lamp Types	Displays the lamp type (lamp specification or ANSI code), as set in the default parameters of the symbol (in the Light Info Record)
Show Counts	Displays a count value for each instrument type
Compare to Inventory	Compares and displays the counts against the total number of units of each type as calculated in the inventory (see "Lighting Inventory Setup" on page 960)
Left Margin	Sets the maximum distance to the left of the summary insertion point for the symbols to display, in page units
Item Spacing	Specifies the vertical distance between instrument graphics, in page units. Additional distance is automatically added if the symbol is larger than the spacing.
Text Distance	Sets the distance between the summary insertion point and the text origin, in page units
Text Width	Sets the maximum width of text before it wraps, in page units
Headers	
Section Align	Aligns the list header to the left or center

Parameter	Description
Size/Style	Specifies the text formatting for the header
Color Summary	
Show Swatch	When a color summary is included, adds a color swatch for each color used in the drawing
Border	
Style	Select the style of border for the summary, or whether to omit the border
Separation	For the double border style, sets the distance between the lines
Padding	The distance, in page units, between the bounding box of the summary and the border
Title	
Show Title	Displays a title for the instrument summary
Text	Specify the title text to display
Position	Select where to locate the summary title with respect to the border
Font/Size/Style	Specifies the text formatting for the title

3. Click **Build List** to select the elements to be included in the summary.

   The Build Summary List dialog box opens, to specify what is listed in the summary. You can add instrument symbols in use, instrument symbols present in the file (but not currently in the drawing), and formatting elements such as column headers, notes, and circuit summaries.

   Click on each type of component and move the desired elements to the **Component List**. To change the order, click in the first column and drag the item to the desired position in the list, or click **Sort Symbols** to sort them alphabetically.



Click to show/hide the parameters.

Parameter	Description
Available Components	Swaps the display of available components among used instruments, unused instruments, and other summary items
Used Instrument Symbols	Lists all symbols that are currently in use by a lighting device
Unused Instrument Symbols	Lists all the symbols in the file that are currently not in use by a lighting device (unused symbols display in blue so they can be distinguished from the used symbols in the **Component List**)
Other Elements	Lists summary and display elements which can be added to customize the summary; certain items require the specification of additional customization options after adding them to the **Component List** • **Column break**: Begins a new column in the summary • **Column Headers**: Adds a "Symbol and Description" header to each column • **Begin subtotal**: Indicates the start of a subtotal count section • **End subtotal**: Indicates the end of a subtotal count section; the subtotal value is placed at this location. If a subtotal cannot be calculated, nothing is displayed in the summary. • **Header**: Adds a section header • **Divider**: Places a horizontal separator across the column; separator does not extend into the border padding area • **Full divider**: Places a horizontal separator across the column; separator extends to the border • **Note**: Adds text that is aligned with the symbol text • **Typical symbol**: Places a selected symbol that is normally the reference symbol used for label legends • **Color summary**: Adds a summary of all the gel colors used, with color, gel size, and counts (select **Show Swatch** in the Instrument Summary Settings dialog box or Object Info palette to include a color swatch) • **Circuit summary**: Adds a summary of the circuits in use (usually only included with lighting position summaries) • **Position Height:** Displays the position height value, when the instrument summary is filtered by position. (When not filtered by position, or when position heights vary, the height value displays "Varies.")
Available Components list	Lists the items that can be added to the instrument summary; the available items depend on the selection in **Available Components**. Sort the symbols in ascending or descending order by clicking in the first column header.
Display Components	For symbols which consist of nested symbols, adds the nested symbols to the list

Parameter	Description
>	Adds one or more selected item(s) from the **Available Components** list to the **Component List**. The items are added to the row above the selected row, or to the end of the list if nothing is selected. Alternatively, double-click to add the item(s).
>>	Adds all the items displayed in the **Available Components** list to the **Component List**.
<	Removes one or more selected item(s) from the **Component List**
<<	Removes all the items from the **Component List**
Component List	Lists all the components, in order, to include in the instrument summary. Change the order by clicking in the first column and dragging to the desired location; sort in ascending or descending order by clicking in the first column header. The **Circuits** column displays specified circuit counts for multi-circuit instruments. Click in the **NR** column of a selected symbol to prevent the symbol from automatically rotating in the summary thumbnail display.
Number of Circuits	Select multi-circuit instruments in the **Component List,** and enter the number of circuits used (these are displayed in the **Circuits** column). Rows without a specific number of circuits assume that one circuit is used.
Sort Symbols	Click to sort all the components, including "other elements," in alphabetical order
Options	For "other elements," opens a dialog box with special formatting options

4. If you have included other elements that require formatting, select each one from the **Component List** and click **Options**.
   - For headers, position height headers, or notes, enter the header or note text.
   - For typical symbol, select the symbol to indicate labels (usually a label legend symbol, located within the Label Legend symbol folder)
   - For circuit summaries, the Circuit Summary Options dialog box opens to specify the circuit information to include in a lighting position summary. Specify the types of devices to count and how to count them, and enter the device amperage rating/voltage to obtain a report indicating the circuit type you will need for those devices.

Click to show/hide the parameters.

Parameter	Description
Header	Enter the circuit summary header text
Count By	Select whether to count the circuits by channel, dimmer, or circuit number
Twofer By	Select the method for counting two-fered circuits as one circuit: • **None**: Count all circuits, including ganged circuits • **Explicit**: Count only two-fer circuits that were created by the **Ganging** tool • **Anywhere**: Counts any two-fered circuits, including those created by the **Ganging** tool and any set of two or more lights that are on the same circuit

Parameter	Description
Amperage Ratings	Specify the amperage ratings for the lighting devices, separating multiple values with a semi-colon (for example, *20;50* indicates 20A and 50A). Amperage rating is determined by a combination of a device's wattage and voltage. Some examples include: • With a voltage of 120V and an instrument with a 5000W rating, the instrument would go into the 50A circuit • With a voltage of 120V and an instrument with a 500W rating, the instrument would go into the 20A circuit • With a voltage of 120V and three 1000W instruments two-fered together, the instruments would go into the 50A circuit with two 50A two-fers
Voltage	Specify the circuit voltage
Device Types	Select the types of devices to count in the circuit summary; typically two or more summaries are included, combining selected device types and specific power requirements so that you know the amperage required per circuit for various device types

5. Click **OK** to return to the Build Summary List; continue specifying summary list options.
6. Click **OK** to close the Build Summary List dialog box and return to the Instrument Summary Settings dialog box.
7. Click **OK**.
8. Click in the drawing to place the instrument summary. Click again to set the rotation.

   The instrument summary parameters can be edited from the Object Info palette. However, the **Summary Defaults for** option is not available since the summary already exists for instruments or lighting positions.

   Filtering and refreshing are options that are available from the Object Info palette after an instrument summary has been added to the drawing.

Click to show/hide the parameters.

Parameter	Description
Active Filters	Displays the status of any summary filtering in effect
Filters	Opens the Summary Filters dialog box, for specifying filtering of the summary items by lighting position, layer, class, or custom criteria
Refresh	When instruments in the drawing, lighting positions, inventory counts, or other factors have changed, updates the instrument summary with the latest data

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating an Instrument Summary for a Lighting Position
Filtering Instrument Summary Contents
Creating Instrument Summaries

Creating an Instrument Summary for a Lighting Position

An instrument summary can be created for a lighting position rather than for the entire drawing.

To create an instrument summary for a selected lighting position:

1. Select the lighting position.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. From the Object Info palette, click **Insert Position Summary**.
3. A position summary is added next to the lighting position.

   If a summary build list has never been specified in the file, a placeholder labeled **Build List** is placed instead, indicating that a build list has not yet been specified in the file and needs to be set.

   The position summary uses the **Summary Defaults for** parameters for a position, as set up in the instrument summary settings, and the summary is filtered for the current lighting position. As instruments are added to the position, click **Insert Position Summary** to add the instruments to the end of the summary. To change specific instrument summary parameters, select the instrument summary and edit the parameters in the Object Info palette as described in "Creating an Instrument Summary" on page 961.

   If you delete a lighting position, the associated instrument summary is also deleted.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Lighting Positions
Creating an Instrument Summary
Filtering Instrument Summary Contents
Creating Instrument Summaries

Filtering Instrument Summary Contents

Instrument summaries can be filtered to limit the contents by lighting position, layer, class, or specified criteria.

To filter the instrument summary contents:

1. Select an instrument summary.
2. From the Object Info palette, click **Filters**.

   The Summary Filters dialog box opens. Each tab includes filtering options to limit the summary contents according to specified criteria. The selections made on each tab are additive.

Click to show/hide the parameters.

Parameter	Description
Position	Click in the first column to select one or more lighting positions to use as a filter; instruments from lighting positions with a check mark are included Add a **Position Height** header to the summary elements to include the position height for each lighting position.
Layers	Click in the first column to select one or more layers to use as a filter; instruments from layers with a check mark are included
Classes	Click in the first column to select one or more classes to use as a filter; instruments from classes with a check mark are included
Custom	Specify custom filtering criteria by entering a criteria string, or by clicking **Criteria** and specifying the exact filtering criteria in the Criteria dialog box (which automatically creates the criteria string). For more information on specifying criteria, see "Entering Formulas in Worksheet Cells" on page 1335.

3. Click **OK**.

   The summary list is filtered. The **Active Filters** in the Object Info palette displays the current filters and/or criteria string.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating an Instrument Summary
Creating an Instrument Summary for a Lighting Position
Creating Instrument Summaries
Inserting Lighting Positions

Generating Paperwork

The information to be included in the various schedules and reports, as well as the format of the reports, is specified in the Generate Paperwork dialog box.

> The **Choose Schedule** command automatically creates certain reports as preformatted worksheets; see “Creating Schedules Automatically” on page 1318.

To set up the schedule and report information:

1. Select **Spotlight > Reports > Generate Paperwork**.

   The Generate Paperwork dialog box opens.

2. Select the desired **Schedules** and **Reports**. Each schedule and report selected must be set up by clicking on its **Setup** button. The setup procedure is described in the following sections.

3. Configure the rest of the paperwork setup by entering the **Header Configuration**, **Show Information** and the **Page** properties. Click **OK** to generate the desired paperwork with the specified settings.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Schedule Setup
Inventory Reports
Magic Sheets
Color Cut List
Header Configuration
Show Information
Page Properties
Reviewing Generated Paperwork
Creating Schedules Automatically

Schedule Setup

The information to be included in the various schedules, as well as the format of the schedules, is specified in the Schedule Formatting dialog box.

To set up the schedule contents and formatting:

1. Select **Spotlight > Reports > Generate Paperwork**. The Generate Paperwork dialog box opens.
2. Click the **Setup** button under the Schedules list. The Schedule Formatting dialog box opens.

   Each schedule in the **Schedule** list is formatted in the same way. The formatting must be specified for each schedule by first selecting the schedule from the list and then specifying the format.

   When selecting **Available fields** and **Column Order**, press and hold the Shift key to select multiple, contiguous items or press and hold the Ctrl key (Windows) or Command key (Mac) to select non-contiguous items.

Click to show/hide the parameters.

Parameter/Button	Description
Printed Name	Suggests a title for the top of the report of the selected schedule. Change the title, if desired, by typing in a new title for that schedule.
Available Fields	Displays the items that can be included in the schedule
Schedule Columns	Displays, in order, the columns that have been selected to appear in the schedule
Move >	Adds a selected available item to the list of schedule columns
Move <	Removes a column from the **Schedule Columns** list
Move Up/Move Down	The most recently added item displays at the bottom of the **Schedule Columns** list, indicating that it is the last column in the report; to change the column order, select a column and click on the **Move Up** and **Move Down** buttons until it is in the desired order
Column Format	For each column in the **Schedule Columns** list, select **Wide**, **Medium**, or **Narrow** from the **Width** list. These selections represent the relative widths of the columns in relation to the other columns. Since the schedule is automatically fit to the page, specifying the actual widths is unnecessary. The actual width of the column is dependent upon the number of columns as well as the final printed schedule size.
Page Formatting	Specifies the formatting of the Column Headers and report Body text; select the **Font**, **Style**, and **Size** for each. Click the **Change** button to open the Format Text dialog box for editing text parameters (see “Formatting Text” on page 389).

3. After specifying the column information, order, and width, as well as the page formatting for each type of schedule to be generated, click **OK** to return to the Generate Paperwork dialog box. The schedule formatting is used when the paperwork is generated.

   The Schedule formatting settings are saved in the file’s Schedule Formats worksheet. The worksheet, and all the formatting, can be imported into another file through the Resource Browser.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Generating Paperwork

Inventory Reports

An inventory report, showing the items currently placed on the light plot and the items available, can be included in the generated paperwork. The inventory report can be set up from the Generate Paperwork dialog box, or from the **Spotlight > Reports > Lighting Inventory Setup** command.

To include an inventory report:

1. Select **Spotlight > Reports > Generate Paperwork**, and then select **Inventory** from the Generate Paperwork dialog box. To create an inventory report that will separate instrument information according to lighting position, select **Break Down by Position**.
2. Click the **Setup** button next to **Inventory**. The Lighting Inventory Setup dialog box opens.
3. Select each device type row, and enter the number of available instruments in the **Quantity**.

   The devices can be sorted by clicking in the **Type** column header.
4. Click **OK** to close the Lighting Inventory Setup dialog box and return to the Generate Paperwork dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Inventory Setup
Generating Paperwork

Magic Sheets

Magic sheets are graphical representations of instruments on the plot; they can be broken down either by focus point or color. Magic sheets are invaluable for cueing a show. They are simple to set up and generate in the Vectorworks Spotlight product.

To create a magic sheet:

1. Select **Spotlight > Reports > Generate Paperwork**, and then select **Magic Sheets** in the Generate Paperwork dialog box.
2. Click the **Setup** button next to **Magic Sheets**. The Magic Sheet Setup dialog box opens. Specify the magic sheet setup parameters.

Action	Description
Generates the magic sheets with lighting summarized by color	Click **Color**; each instrument of a particular color is shown in one view. The general **Theatre Style** needs to be selected from the list to be used as a reference for the color summaries.
Generates the magic sheets with lighting summarized by focus area	Click **Area**. All instrumentation that has a specified focus point will be included in the report. If desired, the magic sheet color number value can be displayed in an approximation of its gel color. Select **Show Color Name in color**.

   The Theatre Styles are editable symbols. Custom styles can be added into the file's Theater Types folder using the Resource Browser.

3. Click **OK** to close the Magic Sheet Setup dialog box and return to the Generate Paperwork dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Generating Paperwork

Color Cut List

A color cut list in Vectorworks shows all the colors required by the show, and calculates the number of each cut size required per color.

To create a color cut list:

1. Select **Spotlight > Reports > Generate Paperwork** and then select **Color Cut List** in the Generate Paperwork dialog box.
2. If desired, select **Break Down by Position** to further break the color cut list down by lighting position.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Generating Paperwork

Header Configuration

In the Generate Paperwork dialog box, select the position of the header information specified in **Show Information**. For the **Left**, **Center**, and **Right** header positions, select the information to be displayed from the list (Designer, Show Name, and/or Date). If no header information is desired for the location, select **None**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Generating Paperwork
Show Information
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Show Information

In the Generate Paperwork dialog box, enter the name of the lighting **Designer** and the **Show Name**, if desired. Type the date of the show or event. To use today's date, click **Date**. Select the show information's position in the header in **Header Configuration**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Generating Paperwork
Header Configuration

Page Properties

In the Generate Paperwork dialog box, enter the **Page Height** and **Page Width** values in the default file units. These values specify the print area (page width with printer margins). The width of the schedule columns is affected by this value.

If desired, a different unit can be used by specifying the value and the unit. For example, even if the default file units are centimeters, specify eight inches for the Page Properties by entering 8". Click **OK** to generate the selected schedules and reports in the format specified.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Generating Paperwork

Reviewing Generated Paperwork

Once the paperwork has been formatted and generated, it can be checked and printed.

Generated Magic Sheets

If Magic Sheets were generated, a new design layer is added to the file. Depending on the type of Magic Sheet generated, the new design layer is called **Magic Sheet-Area** or **Magic Sheet-Color**. The magic sheets are placed on this design layer, and formatted to the specified page size (select **View > Zoom > Fit to Objects** if the Magic Sheet items cannot be seen). If the light plot parameters are changed, the magic sheets must be regenerated to reflect the updates.

Paperwork Worksheets

The specified reports and schedules, once generated, display in the Resource Browser under the Worksheets heading of the current file.

For more information on worksheets, see "Worksheets" on page 1315.

Circuit Hookup @ 100%

File Edit View Insert Format

A1

	A	B	C	D	E	F	G	H	I
1	IJD				As You Like It				Wednes
2					Circuit Hookup				
3									
4	Circuit	Dimme	Channe	Instrument Type	Wattage	Purpose	Color	Position	Unit N
5	1	1	111	Alt 360Q 6x12	750W		R54	Balcony Rail	9
6	1	1	128	Alt 360Q 6x12	750W		R54	Balcony Rail	5
7	2	2	113	Alt 360Q 6x12	750W		R54	Balcony Rail	10
8	2	2	120	Alt 360Q 6x12	750W		R54	Balcony Rail	4
9	2	2	126	Alt 360Q 6x12	750W		R54	Balcony Rail	7
10	3	3	115	Alt 360Q 6x12	750W		R54	Balcony Rail	12
11	4	4	122	Alt 360Q 6x12	750W		R54	Balcony Rail	2
12	5	5	116	Alt 360Q 6x12	750W		R54	Balcony Rail	11
13	6	6	112	Alt 360Q 6x12	750W		R54	Balcony Rail	8
14	6	6	127	Alt 360Q 6x12	750W		R54	Balcony Rail	6

The following points should be remembered when working with generated paperwork:

- Edits to this type of worksheet do not update the light plot parameters.
- If the light plot parameters are edited, this type of report or schedule must be regenerated to reflect the updates.

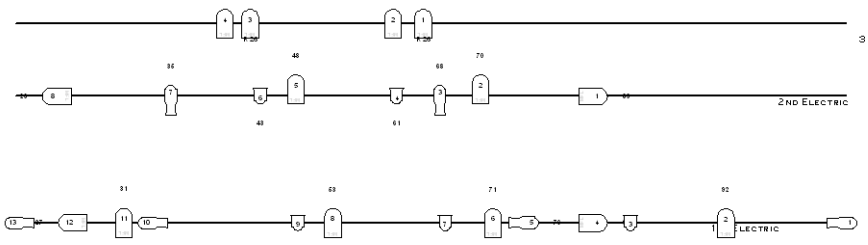
Creating Hanging Cards

Lighting position hanging cards can be created.

To create hanging cards:

1. Set the design layer visibility by selecting **Window > Palettes > Navigation** and clicking the Design Layers tab. Set only the design layers with the desired lighting positions and instruments to visible.
2. Similarly, set class visibility by clicking the Classes tab and setting only the desired classes to visible.
3. Select **View > Create Viewport**.

   The Create Viewport dialog box opens. For more information on viewports, see “Creating Sheet Layer Viewports” on page 1616.
4. From the **Create on Layer** field, select the sheet layer for the viewport (or create a new sheet layer).
5. Select **Display Planar Objects** and **Project Screen Objects**, with Top/Plan view and Wireframe rendering. Click **OK**.
6. The viewport is created and the sheet layer displays.

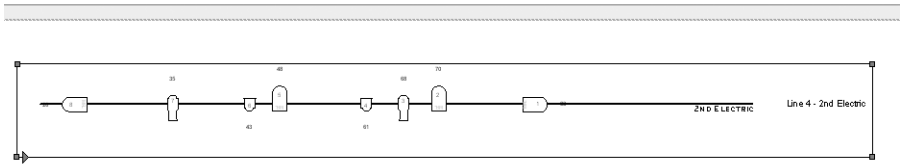


7. Select **Modify > Edit Viewport**.

   The Edit Viewport dialog box opens. Select **Crop** and click **OK**.

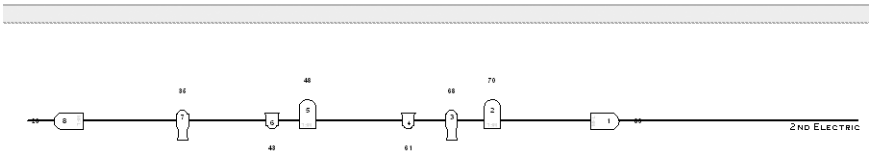
   Alternatively, right-click (Windows) or Ctrl-click (Mac) the viewport and select **Edit Crop** from the context menu to crop the viewport.

8. In crop viewport mode, draw a 2D shape to crop the desired lighting position.



   To hide the cropping shape, set its line thickness to zero in the Attributes palette. Click **Exit Viewport Crop** at the top right of the drawing window to crop the viewport.

9. The cropped viewport displays on the sheet layer.



10. Several viewports can be created on the same sheet layer to display all the desired hanging cards. Changes to the design layer(s) are automatically reflected in the viewport.
11. Add lighting position summaries to the sheet layer as described in “Creating an Instrument Summary for a Lighting Position” on page 966; adjust the **Scale** of the displayed symbols so that they do not display at a 1:1 scale, but are sized appropriately for the sheet layer.

S R Inserting Gobo Projections

A pattern, texture, or color can be placed on a lighting instrument and projected onto the stage in. This special light source and instrument combination is a gobo projection.

Lighting instruments with gobo projections cast a light beam that can be rendered by the Renderworks product. You can then preview the effect of the gobo projection texture(s) on the stage.

The Renderworks product must be installed to render gobo projections. For more information on rendering, see “Rendering the Drawing” on page 1569.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Gobo Texture Libraries
Creating Gobo Textures
Editing Gobo Texture Transparency Settings
Inserting a Gobo Projector
Showing Gobo Projections

S Gobo Texture Libraries

The Vectorworks Spotlight product includes thousands of commercial gobo textures from Apollo, Lee, GAM, GOBOLAND, High End Systems, and Rosco. Through the Resource Browser, import the texture into the current file from one of the files located in the gobo textures folder of the [Vectorworks]\Libraries folder that is included with the Vectorworks Spotlight product (see “Resource Libraries” on page 219). To use the texture in a gobo projection, specify its name in the Object Info palette (see “Creating Gobo Textures” on page 974).

When the Vectorworks Spotlight product is installed, gobo textures are also provided as default content (default content is automatically imported into the file when selected while changing instrument properties, and displays in the Resource Browser; see “Editing Lighting Instruments” on page 877, and “Resource Libraries” on page 219.)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

S Inserting a Gobo Projector

A gobo projector is a lighting instrument with a light source and gobo projector specified.

1. Insert a lighting instrument as described in “Inserting Instruments” on page 875.
2. In the Object Info palette of the selected instrument, click **Edit**.

   The Lighting Device dialog box opens. This provides a convenient way of specifying parameters (see “Changing Instrument Properties” on page 880), although the parameters can also be entered directly in the Object Info palette.
3. On the Instrument Properties tab, specify a focus point for the instrument.
4. On the Light Information tab, for **Gobo 1**, enter the name of the gobo projection texture (if it has been imported into the file as a resource), or click **Get Resource** to select a gobo texture from the default content. Specify the **Gobo 1 Rotation** angle, if any.
5. If there is a second gobo texture, specify its parameters in **Gobo 2**.
6. Click **OK**.
7. Turn on the embedded light source for the lighting instrument.

   Either right-click on the light, and select **Edit Light** from the context menu to open the Properties dialog box (and click **On**), or turn on the light from the Visualization palette.

   To preview the effect of a color projection on the stage, indicate the **Color** name in the Object Info palette of a selected instrument, without specifying a gobo texture. Render the gobo projection to project the light on the focus point with the specified color. The colors can be selected from the color libraries installed with the Vectorworks Spotlight product. The color code must be entered in a “Manufacturer color value” format (for example, R101). If the color value cannot be found, the color defaults to white.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Changing Instrument Properties
Lighting Instrument Properties
Creating Gobo Textures
Editing Gobo Texture Transparency Settings
Showing Gobo Projections

S R Creating Gobo Textures

In addition to the gobo images available in the pre-defined commercial gobo projection libraries, any square image can be converted into a gobo projection texture.

The Renderworks product must be installed in order to create gobo textures.

Most image-based textures are automatically compressed when imported into a Vectorworks file. Imported JPEG files retain the original JPEG data; all other image files are compressed using lossless PNG format.

To create a gobo texture:

1. Select **Spotlight > Visualization > Create Gobo Texture**.

   The Create Gobo Texture dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Enter a name for this texture	Specifies the gobo texture name
Edit Texture	Edits the gobo texture transparency settings at creation. Once an image has been selected (as described in the following steps), the Edit Texture dialog box opens for editing texture transparency settings.

For more information on editing textures, see “Editing Textures and Shaders” on page 1516.

2. Click **OK.**

   If a resource with an image is already present in the file, the Choose Image dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Import an Image File	Imports a new image. Click **OK** and proceed to Step 3.
Reuse an Image from Another Resource	Reuses a previously imported image; select the resource that contains the image. Click **OK** and proceed to Step 4.

3. Select the desired image file and click **Open**.
4. If **Edit Texture** was selected in Step 1, the Edit Texture dialog box opens. Select the desired options and click **OK**. Click **OK** again to close the Create Gobo Texture dialog box.

   The texture resource is created and is listed in the Resource Browser.
5. Associate a defined texture with an instrument by entering the texture name in the **Gobo 1** or **Gobo 2** field of a selected instrument’s Object Info palette, and specify the **Gobo Rotation**, if any.

   A texture can be saved in the default content file located in gobo textures folder of the [Vectorworks]\Libraries folder that is included with the Vectorworks Spotlight product (see “Resource Libraries” on page 219). Import the texture into the default file, or save a new default file within the folder.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Gobo Texture Transparency Settings
Inserting a Gobo Projector
Showing Gobo Projections

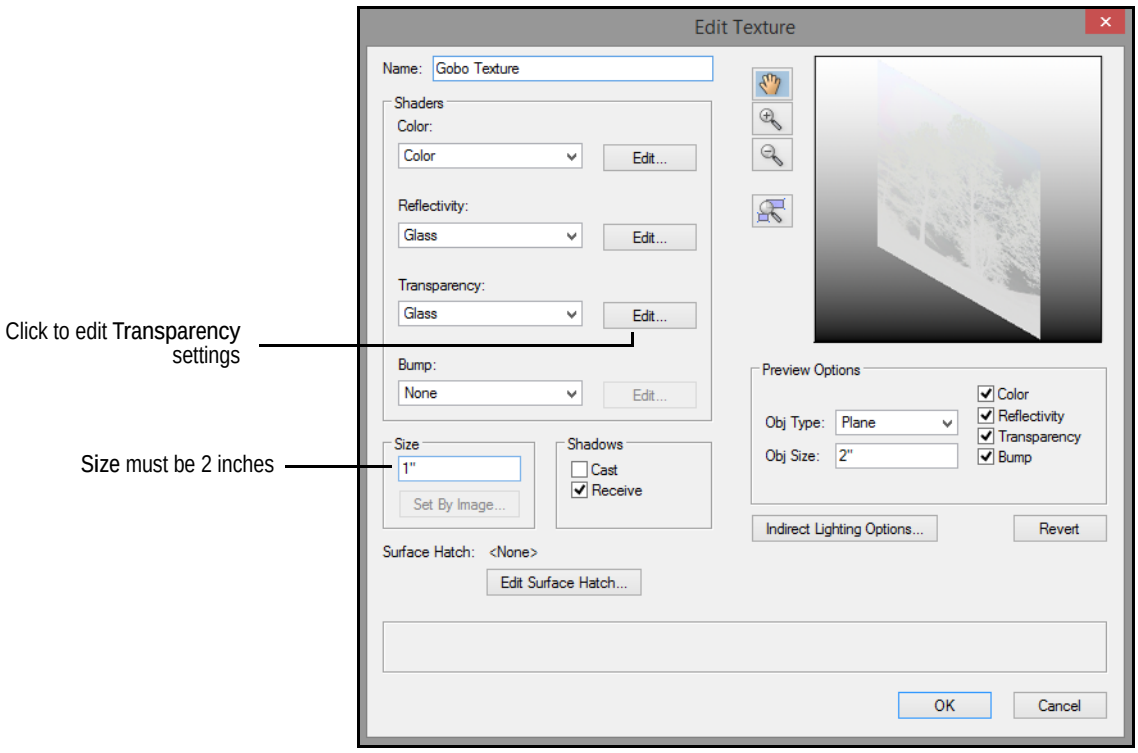
S R Editing Gobo Texture Transparency Settings

A gobo texture is a transparent image resource listed in the Resource Browser. The transparency settings can be edited after the gobo image has been specified, or at creation.

To edit a created gobo texture’s transparency:

1. Select the resource in the Resource Browser.
2. Select **Resources > Edit** to open the Edit Texture dialog box. The Image Transparency settings can be edited. For more information on editing textures, see “Editing Textures and Shaders” on page 1516.

   The texture **Size** must be 2 inches.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Creating Gobo Textures
Inserting a Gobo Projector
Showing Gobo Projections

S R Showing Gobo Projections

The gobo textures and color of a selected light instrument can be projected in a rendered simulation. This rendered image allows you to preview the effect of the gobo texture.

Gobo Projection Requirements

To be able to project a gobo texture in Vectorworks Spotlight, the lighting instrument must:

- Have one or two gobo textures specified
- Be aimed at an existing focus area specified in the **Focus** field of the Object Info palette
- Have its associated light source turned on (either right-click on the light, and select **Turn On** from the context menu, or turn on the light from the Visualization palette)
- Have **Draw Beam** deselected in the Object Info palette

The following Vectorworks software requirements must be met:

- The design layer containing the instrument with gobo texture must also contain 3D geometry (any 3D object that is capable of accepting a Renderworks light) to project the texture
- The Renderworks product must be installed
- Either Fast Renderworks or Custom Renderworks can be used

  For Custom Renderworks, at a minimum, **Shadows** and **Textures** must be selected in the custom Renderworks settings.

To select the custom settings, select **View > Rendering > Custom Renderworks Options**. For more information on Custom Renderworks settings, see “Custom Renderworks Options” on page 1600.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Projecting a Gobo Texture
Creating Gobo Textures
Editing Gobo Texture Transparency Settings
Inserting a Gobo Projector

S R Projecting a Gobo Texture

To project a gobo texture:

1. Make all the settings and meet all the requirements described in “Inserting a Gobo Projector” on page 974, and “Gobo Projection Requirements” on page 976.
2. Select **View > Rendering > Custom Renderworks**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Gobo Textures
Editing Gobo Texture Transparency Settings
Inserting a Gobo Projector
Gobo Projection Requirements

S Managing Scenes

The levels, colors, positions, and focusing of all the instrument objects can be saved as a lighting scene. The scenes can then be used to create movies of the scene transitions.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Saving Scenes
Animating Scenes

S Saving Scenes

To save a scene:

1. After the lighting properties for all the instruments have been set correctly, select **Spotlight > Visualization > Manage Scenes**. The Manage Scenes dialog box opens.
2. In the **Scenes** list, the list of saved scenes is displayed. The scenes are sorted by the order in which they are entered. To save the current settings as a scene, click **Save**. The Save dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Scene Name	Specifies the scene name
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Scene Number	Specifies the scene number
Up Time	Enter the time allotted for the lights to increase in intensity, in seconds
Down Time	Enter the time allotted for the lights to decrease in intensity, in seconds

3. Click **Done** to save the scene. It is added to the **Scenes** list.

Editing Scenes

To edit a scene:

1. Select the scene from the list in the Manage Scenes dialog box and click **Edit**.
2. Make the desired change to the **Scene Name** or **Number**, or the **Up/Down Time**, then click **Done** to save the change. Confirm the changes.

Removing Scenes

To remove a scene:

1. Select the scene from the list in the Manage Scenes dialog box and click **Remove**.
2. Confirm that you wish to remove the scene, then click **Yes**. The scene is removed from the **Scenes** list.

Restoring Scene Lighting Parameters

To restore lighting parameters from a scene:

1. Select the scene from the list in the Manage Scenes dialog box and click **Go!**.
2. Confirm that the parameters are to be restored. The current light parameters are replaced by those from the scene.

   All current levels, colors, positions, and focus points for all the instruments will be replaced. Any information that has not been saved will be lost.

S R Animating Scenes

In Vectorworks, a movie of scene settings can be created to preview the cue transition between the scenes. The contents of the active window, exactly as they display, are used to create the movie. Therefore, the images must be rendered before animating the scenes to accurately preview the lighting effect. These features cannot be used to accurately render all the lighting instruments on the stage for every scene.

To animate the scenes:

1. Set the rendering option parameters described in “Gobo Projection Requirements” on page 976.
2. Save the scenes as described in “Managing Scenes” on page 977.
3. Select **Spotlight > Visualization > Animate Scenes**.
4. In the Choose Path dialog box, select a location and file name for the movie. Click **Save** to open the Animate Scenes dialog box.
5. Enter the parameters for the **Start** and **End Scenes**. Select the scene name from the list; its parameters display underneath.
6. If desired, the scene can be edited by clicking the **Edit Scene** button. Change the **Scene Name**, **Scene Number**, **Up Time** and **Down Time**, and then click **OK** to confirm the changes. The **Up Time** and **Down Time** used to animate the scenes is taken from the End Scene.
7. To create the animation, enter the scene **Hold for** time in seconds for both the starting and ending scenes.

8. Edit the movie settings by clicking the **Compression Settings** button. The Compression Settings dialog box opens.
9. Enter the **Frames per second** and **Quality** for the movie file and click **OK**.
10. When the scene animation settings are complete, click **OK**. The progress is displayed on the screen as the movie is created in the specified location, based on the settings that were entered.
11. To see the movie, locate the movie file and double-click to play it.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Managing Scenes
Gobo Projection Requirements

Creating Plot and Model Views

The **Create Plot and Model View** command automatically creates one or more design layer viewports of geometry to be viewed and rotated in a model layer. This allows users to create both a 2D lighting plan and model views with unique rotation angles, from the same geometry (lighting positions, trusses, lighting instruments, and associated geometry). This command represents a lighting design with non-horizontal lighting positions in Top/Plan view and a properly represented model in the model layer.

Model views can be created in two ways. If the command is run when no objects are selected, special "definition layers" are created for each instance of a lighting position, as well as the remaining non-architectural geometry, which is placed in another definition layer. The associated design layer viewports on the model layer can be rotated individually for that geometry, with great flexibility, or automatically set to a vertical orientation. Alternatively, run the command with one or more Vectorworks Spotlight objects selected; a single design layer viewport is created on the model layer, and its rotation can be automatically set to vertical if desired. This is useful, for example, when selected trusses need to be viewed in the model with the same vertical orientation.

Architectural geometry, such as floors and walls, cannot be rotated in the model layer.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Model Views of the Entire Plot
Creating Model Views of Selected Geometry

Creating Model Views of the Entire Plot

To create model views from all valid geometry in the plot layer:

1. From a plot layer that contains valid Vectorworks Spotlight objects (lighting positions, trusses, lighting instruments, and non-architectural 2D or 3D geometry or hybrid symbols), ensure that no items are selected.
2. Select **Spotlight > Visualization > Create Plot and Model View**.

   The Create Plot and Model View dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies a name for the definition layer(s); "Definition Layer" is the default name. Additional layers are based on the name set here, with an automatically incremented and appended number (Definition Layer-1, Definition Layer-2, and so on).
Model Layer	Specifies where the model layer will be created; select a layer name from the list, or create a new layer. If New Model Layer is selected, you will be prompted to create a design layer after clicking **OK**.

Parameter	Description
Separate	Creates a new definition layer for each valid lighting position and remaining non-architectural geometry, allowing each design layer viewport to rotate independently in the model view; deselect to create a single definition layer with all valid objects in a single design layer viewport
Vertical	Automatically sets all non-architectural geometry to a vertical orientation in the model layer

3. Click **OK**.

   A design layer viewport is created for each lighting position instance and non-architectural geometry, replacing the original objects on the plot layer.

   - If **Separate** was deselected, a single definition layer contains all Vectorworks Spotlight geometry.
   - If **Separate** was selected, a definition layer exists for each instance of the Vectorworks Spotlight geometry.

   Finally, a model layer contains design layer viewports for rotation and proper positioning.

4. In the model layer, use the **Rotate** tool to orient the design layer viewports. See “Rotate Tool” on page 1022.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Plot and Model Views
Creating Model Views of Selected Geometry

Creating Model Views of Selected Geometry

To create model views from selected valid geometry in the plot layer:

1. From a plot layer that contains valid Vectorworks Spotlight objects (lighting positions, trusses, lighting instruments, and non-architectural 2D or 3D geometry or hybrid symbols), select the items to be rotated in a model view.
2. Select **Spotlight > Visualization > Create Plot and Model View**.

   The Create Plot and Model View dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies a name for the definition layer; “Definition Layer” is the default name.
Model Layer	Specifies where the model layer will be created; select a layer name from the list, or create a new layer. If New Model Layer is selected, you will be prompted to create a design layer after clicking **OK**.
Separate	This option is disabled when drawing items are selected
Vertical	Automatically sets the selected item(s) to a vertical orientation in the model layer

3. Click **OK**.

   A design layer viewport is created for the selected Vectorworks Spotlight geometry, replacing those objects on the plot layer; if **Vertical** was selected, the model view displays the design layer viewport in a vertical orientation. A single definition layer contains the original selected Vectorworks Spotlight geometry.

4. If **Vertical** was not selected, use the **Rotate** tool in the model layer to orient the design layer viewport. See “Rotate Tool” on page 1022.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Plot and Model Views
Creating Model Views of the Entire Plot
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Event Design

The event design suite available in Vectorworks Spotlight contains automated tools to facilitate setting up a complete room layout for an event such as a banquet, presentation, or training session. Automatically create the room for the event, including the floor, walls, stage, steps, lectern, video screens, and seating. View the room setup in a rendered or plan view, and obtain a report on the total number of tables and chairs required for the event.

The event design suite uses objects to quickly and automatically create the room plan with only a few parameter settings required. However, due to the powerful nature of these objects, they can be specifically tailored through the Object Info palette to customize their appearance if needed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating the Room

The first step of event design is usually creating the event room. Based on a simple 2D shape, the room and floor are created automatically according to specified parameters.

To create the room:

1. Create a closed 2D object (such as a rectangle, polyline, or polygon) to represent the room. The dimensions of the object should match the interior dimensions of the room to be created.
2. With the 2D object selected, select **Event Design > Create Room**.

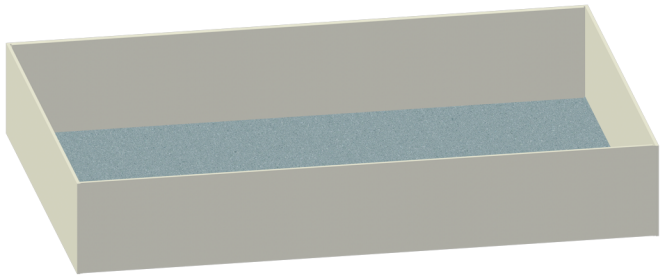
   The Create Room dialog box opens. Specify the properties and attributes of the walls and floor.

Click to show/hide the parameters.

Parameter	Description
Walls	Specify the wall attributes
Height	Sets the wall height of the room's walls
Class	The wall attributes can be set by class, by choosing a class other than None. If desired, select a class from the list of classes in the file, or create a new class for the walls. Leave the walls in the None class to set the remaining attributes from the dialog box.
Thickness	Sets the wall thickness
Fill Hatch	Sets a fill hatch for the walls from the file's resources
Fill Color	Sets the fill color for the walls
Pen Color	Sets the pen color for the walls
Texture (Renderworks required)	Specifies the overall wall texture from either the default content or the current file's content
Floor	Specify the floor attributes
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Class	The floor attributes can be set by class, by choosing a class other than None. If desired, select a class from the list of classes in the file, or create a new class for the floor. Leave the floor in the None class to set the remaining attributes from the dialog box.
Fill Color	Sets the fill color for the floor
Pen Color	Sets the pen color for the floor
Texture (Renderworks required)	Specifies the floor texture from either the default content or the current file's content

3. Click **OK**. The room's walls and floor are automatically created.



Rendered 3D view for clarity

Wall and floor parameters can be edited in the Object Info palette; see "Editing Walls" on page 535 and "Creating Floors" on page 481.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Walls
Creating Floors
Creating the Stage
Creating Stage Stairs
Creating a Lectern
Creating a Video Screen
Creating Event Seating
Creating Event Views
Setting Class Properties
Resource Libraries

Creating the Stage

Many event rooms contain a stage for viewing presentations. Based on the **Stage Plug** tool, a stage is automatically created according to specified parameters.

The structure type of the stage plug is automatically set to Border.

To create the stage:

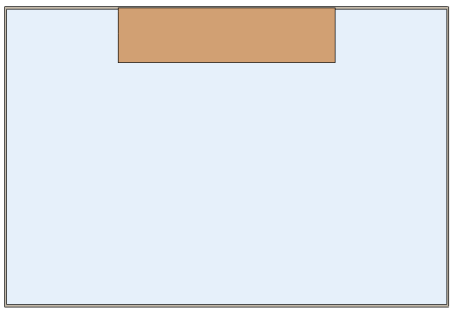
1. Create a closed 2D object (such as a rectangle, polyline, or polygon) to represent the stage. The dimensions of the object should match the dimensions of the stage to be created.
2. With the 2D object selected, select **Event Design > Create Stage**.

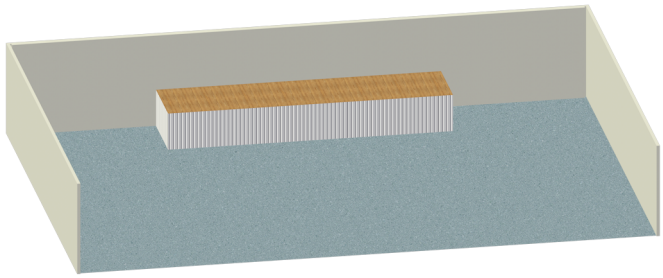
   The Create Stage dialog box opens. Specify the stage properties and attributes.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Click to show/hide the parameters.

Parameter	Description
Height	Sets the height of the stage
Class	The stage attributes can be set by class, by choosing a class other than None. If desired, select a class from the list of classes in the file, or create a new class for the stage. See “Setting Class Properties” on page 179. Leave the stage in the None class to set the remaining attributes from the dialog box.
Fill Color	Sets the fill color for the stage
Pen Color	Sets the pen color for the stage
Top Texture (Renderworks required)	Specifies the texture for the top of the stage from either the default content or the current file’s content
Side Texture (Renderworks required)	Specifies the texture for the side of the stage from either the default content or the current file’s content

3. Click **OK**. The stage is automatically created.





Rendered 3D view for clarity; room wall removed for visualization

Stage parameters can be edited in the Object Info palette; see “Inserting a Stage Plug” on page 946.

Right-click (Windows) or Ctrl-click (Mac) on the stage object and select **Edit** from the context menu. Edit the shape of the object path with the **Reshape** tool.

A stage can also be created by clicking the **Stage Plug** tool from the Spotlight tool set.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating the Room
Creating Stage Stairs
Creating a Lectern
Creating a Video Screen
Creating a Seating Layout
Creating Event Views
Resource Libraries

Creating Stage Stairs

Stairs or steps are usually required to provide access to the stage from the floor. The steps are automatically adjusted to meet the top of the stage. The stage stairs are based on the **Stage Steps** tool.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To add steps:

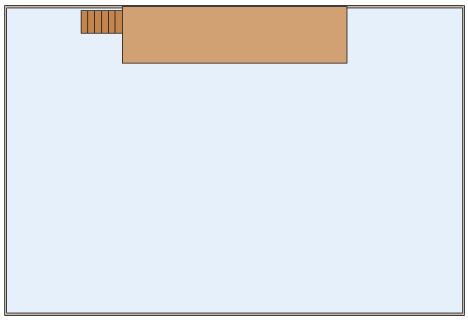
1. Select **Event Design > Create Stair**.

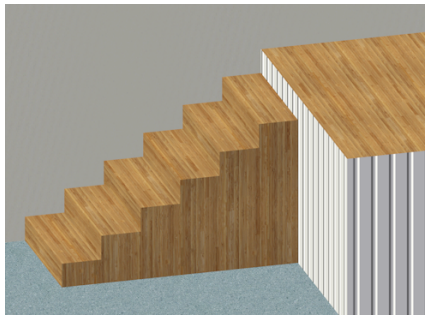
   The Create Stair dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Width	Specify the width of the staircase
Style	Set the type of stair to create: Simple, Self-Adjusting, Frame Free-Standing, Solid Free-Standing, or Open Free-Standing
Class	The stair attributes can be set by class, by choosing a class other than None. If desired, select a class from the list of classes in the file, or create a new class for the stair. See “Setting Class Properties” on page 179. Leave the stair in the None class to set the remaining attributes from the dialog box.
Fill Color	Sets the fill color for the stair
Pen Color	Sets the pen color for the stair
Texture (Renderworks required)	Specifies the stair texture from either the default content or the current file’s content; see “Resource Libraries” on page 219

2. Click **OK**.
3. Click once in the drawing to set the stair position. Click again to set the stair rotation.





Rendered 3D view for clarity

Stair parameters can be edited in the Object Info palette; see “Inserting Stage Steps” on page 949.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Stairs
Creating the Room
Creating the Stage
Creating a Lectern
Creating a Video Screen
Creating Event Seating
Creating Event Views

Creating a Lectern

After the stage and stairs have been added, a lectern can be quickly configured and placed. When placed on the stage, the lectern height is automatically adjusted to the top of the stage.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To add a lectern:

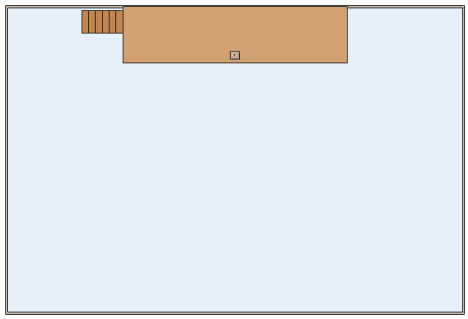
1. Select **Event Design > Create Lectern**.

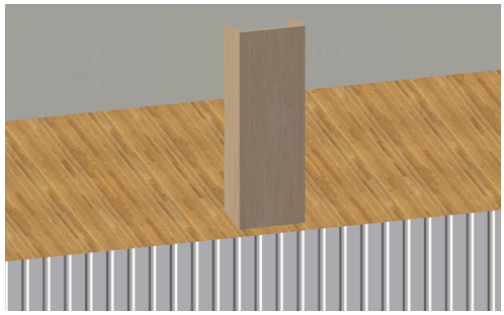
   The Create Lectern dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Choose Lectern	Select one of the lecterns from the list; these are configured with default attributes and can be inserted as is. Alternatively, select a lectern shape to use as the basis for a new lectern symbol with customized attributes.
Custom Attributes	Customizes the selected lectern's default attributes, creating a new hybrid lectern symbol
New Symbol Name	Provide a name for the new lectern symbol; the new symbol is added to the file's resources
Class	The lectern attributes can be set by class, by choosing a class other than None. If desired, select a class from the list of classes in the file, or create a new class for the lectern. See "Setting Class Properties" on page 179. Leave the lectern in the None class to set the remaining attributes from the dialog box.
Fill Color	Set the fill color for the lectern
Pen Color	Set the pen color for the lectern
Texture (Renderworks required)	Specifies the lectern texture from either the default content or the current file's content; see "Resource Libraries" on page 219

2. Click **OK**.
3. Click once in the drawing to set the lectern position. Click again to set the lectern rotation.





Rendered 3D view for clarity

Creating the Room
Creating the Stage
Creating Stage Stairs
Creating a Video Screen
Creating Event Seating
Creating Event Views

Creating a Video Screen

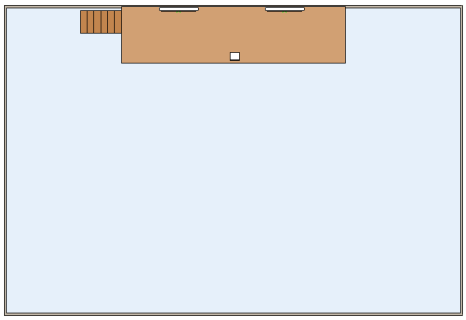
As part of the event design setup, video screens are often necessary at events and can be quickly configured and placed.

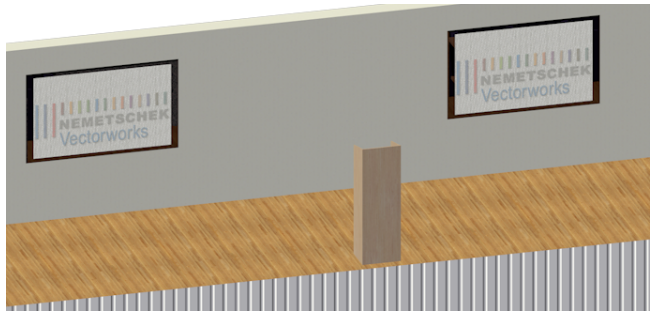
To add a video screen object:

1. Select **Event Design > Create Screen**.

   The Create Screen dialog box opens.

2. Select the type of video screen object to insert, and click **OK**.
3. Click once in the drawing to set the video object position. Click again to set the object's rotation.





Rendered 3D view for clarity

Screen parameters can be edited in the Object Info palette; see "Inserting Video Screen Objects" on page 913.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Video Screen Object
Creating the Room
Creating the Stage
Creating Stage Stairs
Creating a Lectern
Creating Event Seating
Creating Event Views

Creating Event Seating

Once the event room has been defined, one of the last steps is to add seating for the event. A variety of event seating types can be automatically created. Seating can consist of chairs, tables, or tables and chairs. Seating fills a 2D shape in a user-defined arrangement with a "look-to" location defined by a focal point. Along with the event seating, a Seating Count worksheet is created.

To add seating:

1. Create the closed 2D shape (such as a polygon, rectangle, rounded rectangle, oval, arc/circle, or polyline) defining the boundary of the seating area. Objects with one or more holes can be selected; seating will not be placed where a hole exists.
2. With the object(s) selected, select **Event Design > Create Event Seating**.

   The Create Event Seating dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Seating Arrangement	Select the seating spacing and arrangement; any symbol can be selected for the different arrangement types, but normally, banquet seating consists of table and chair symbols, and classroom and theatre seating includes chair symbols. Classroom and theatre seating include an option to arrange the seats concentrically.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Banquet	Typical seating arrangement for banquets, with offset tables spaced equally by the distance set in **Table Spacing**
Classroom	Typical classroom seating arrangement with aligned rows of chairs, spaced according to the distances set in **Seat Spacing** and **Row Spacing**
Theatre	Typical theatre seating arrangement with offset alternate rows of chairs, spaced according to the distances set in **Seat Spacing** and **Row Spacing**
Table Spacing	For banquet seating, indicates the distance between tables
Seat Spacing	For classroom or theatre seating, sets the distance between seats in a row
Row Spacing	For classroom or theatre seating, sets the distance between rows of seats
Concentric	For classroom or theatre seating, specifies a circular layout for the rows
Seating Section Name	Names the seating section for the seating layout worksheet
Seating Symbol	Select one of the seating symbols from the list; these are configured with default attributes and can be inserted as is. Alternatively, select a symbol to use as the basis for a new seating symbol with customized attributes.
Customize Symbol	Customizes the selected seating's default attributes, creating a new hybrid seating symbol

3. Click **Customize Symbol** to change the default appearance of the selected symbol if desired. This creates a new symbol that is added to the file's resources.

   The Customize Symbol dialog box opens.

Click to show/hide the parameters.

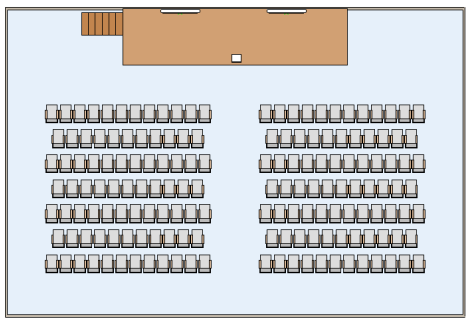
Parameter	Description
Chair/Table Attributes	Sets the attributes for the chair and/or table symbol
Fill Color	Sets the fill color for the chairs and/or tables
Pen Color	Sets the pen color for the chairs and/or tables
Texture (Renderworks required)	Specifies the chair and/or table texture from either the default content or the current file's content; see "Resource Libraries" on page 219
Preview	Displays a preview of the symbol the attributes applied
Render Mode	Select whether to preview the symbol in wireframe or OpenGL rendering mode
New Symbol Name	Provide a name for the new seating symbol; the new symbol is added to the file's resources. Names are automatically generated for nested table and chair symbols.
Class	The seating attributes can be set by class, by choosing a class other than None. If desired, select a class from the list of classes in the file, or create a new class for the symbol. See "Setting Class Properties" on page 179. Leave the seating in the None class to set the attributes from the dialog box.

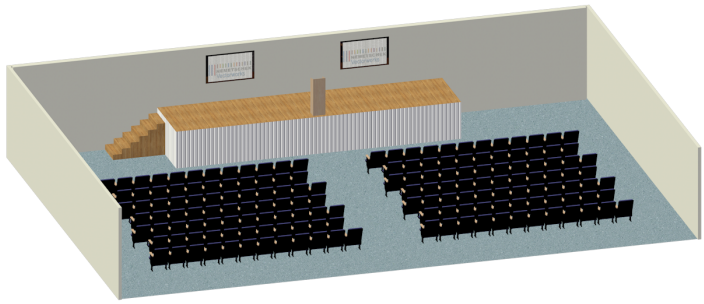
4. Click **OK** to return to the Create Event Seating dialog box.
5. Click **OK** to create the event seating. The Select the Focus Method dialog box opens.

6. Click **Focus at Next Click** to manually place the focal point for the seating, or click **Automatically Focus** to have the focal point created automatically.

   If **Focus at Next Click** was selected, click once to place the focal point.

   All seats will face this point; table and chair symbols orient toward the point. For concentric seat layouts, the distance of the seat focal point from the 2D boundary object determines the radius of the concentric layout. Click close to the object for a smaller radius.





Rendered 3D view for clarity; room wall removed for visualization

The event seating and a Seating Count worksheet are created, with automatically calculated seat and table counts. The Seating Count worksheet displays the number of seats and/or tables required for each seating section, as well as the total number of seats/tables needed for all sections. Nested symbols are counted in the worksheet, so tables and chairs are counted separately within a table and chair symbol. (If a symbol cannot be identified as a table or chair, it is counted as a chair.)

Seating parameters can be edited in the Object Info palette and have the same parameters as the **Create Seating Layout** command (see “Editing the Seating Layout” on page 992).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

A S Creating a Seating Layout

The **Create Seating Layout** command creates a seating layout object from one or more selected polygons, rectangles, rounded rectangles, ovals, arcs/circles, or polylines. Seating can consist of chairs, tables, or tables and chairs. Seating fills the object(s) in a user-defined arrangement with a “look-to” location defined by a control point. Seating layout objects can also be created by drawing a polyline and then selecting the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277). Along with the seating layout, a Seating Count worksheet is created.

To create a seating layout object:

1. Create the shape defining the boundary of the seating area. Objects with one or more holes can be selected; seating will not be placed where a hole exists. Multiple shapes can be selected at one time (to create several seating sections, for example).
2. With the object(s) selected, select the **Create Seating Layout** command from the appropriate menu:
   - Architect workspace: **AEC > Create Seating Layout**
   - Spotlight workspace: **Spotlight > Architectural > Create Seating Layout**
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Create Seating Layout dialog box opens.

Click to show/hide the parameters.

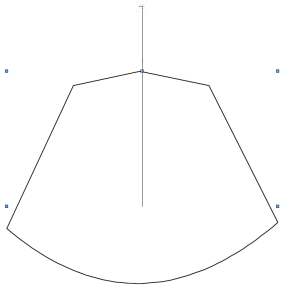
Parameter	Description
Create a new seat symbol called	Creates a new seat symbol; provide the symbol name
Use an existing symbol	Selects an existing seat symbol; specify the symbol from the list
Symbol Folders	Specifies the location of seat symbols
Symbols	Provides a graphical list of available seat symbols

3. A default symbol can be created to represent the seating; select **Create a new seat symbol called** and provide a symbol name. Alternatively, select an appropriate seating symbol from either the default content or the current file's content; see "Resource Libraries" on page 219. In addition to seat symbols, table symbols and table and chair symbols can be used.

   See "Accessing Existing Resources" on page 229 for information on importing symbols into the Resource Browser.

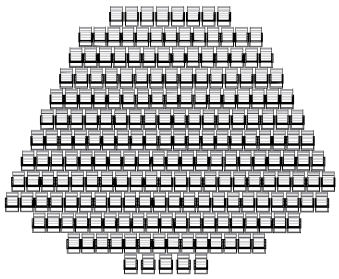
4. Click **OK** to close the Create Seating Layout dialog box.

5. When prompted, click on the focal point for the seating. All seats will face this point; table and chair symbols orient towards the point.



For concentric seat layouts, the distance of the seat focal point from the object determines the radius of the concentric layout. Click close to the object for a smaller radius.

The seating layout object and a Seating Count worksheet are created, with automatically calculated seat and table counts.



Seating Count @ 100%

File Edit View Insert Format

A3 Total:

	A	B	C	D	E
1	**Section Name**	**Seat Name**	**Seat Count**	**Table Name**	**Table Count**
2.1	Outdoor Seating	Bleacher Seat	585		0
2.2	Dining Area	Chair 2	56	Table - 54" Round	7
2.3	Theater Seating	Wooden Seat	397		0
3	**Total:**		**1038**		**7**

The Seating Count worksheet displays the number of seats and/or tables required for each seating layout section, as well as the total number of seats/tables needed for all sections. Nested symbols are counted in the worksheet, so tables and chairs are counted separately within a table and chair symbol. (If a symbol cannot be identified as a table or chair, it is counted as a chair.)

A seating layout can also be created by clicking the **Seating Layout** tool from the Furn/Fixtures tool set. Draw a polygon with the tool and complete the parameters which are identical to those described in "Editing the Seating Layout" on page 992. Alternatively, Spotlight users can use the **Create Event Seating** command on the **Event Design** menu to create a seating layout (see "Creating Event Seating" on page 988).

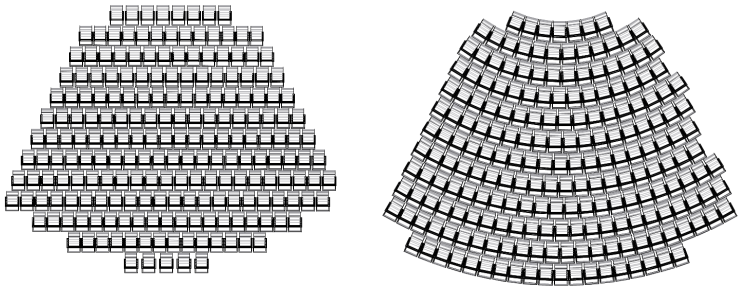
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

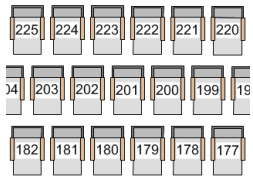
Editing the Seating Layout
Seating Layout Specifications

A S Editing the Seating Layout

Seating layout parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Seat Spacing	Specifies the distance between seats
Row Spacing	Specifies the distance between rows of seats
Section Name	Assigns a section name for use in the Seating Count worksheet
Concentric	Select to draw a concentric seating arrangement  Linear Concentric
Offset Alternate Rows	Offsets alternating rows (starting with the second row), so that the seats are staggered by a half seat width, or chairs and tables are spaced to prevent overlapping
Offset 1st Row	Specifies an offset distance for the first row of seats
Focal Pt X/Y	Indicates the coordinates of the seating look-to point
Focus Front	Click to snap the seating layout's focal point to the front. Because flexible seating areas can have multiple potential fronts, repeated clicks toggle the focal point around the seating area, to each possible front. The focal point snaps to the midpoint of each path segment that defines the seating area.
Draw Boundary Line	Select to display the object used as the basis for the seating layout; deselect to hide the object and show the seats only
Draw Layout Lines Only	Select to display the seating locations without showing the actual seat symbols
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

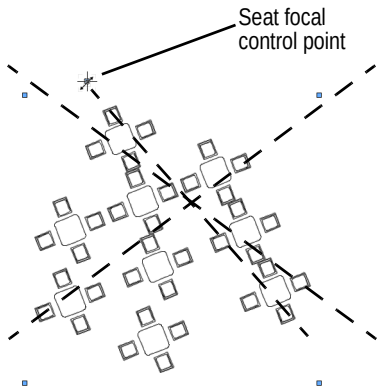
Parameter	Description
Base Z Height	Specifies the initial elevation of the first row of seating
Rise per Row	Specifies the elevation increase for each row of seating (seating layout slope)
Display Seating Section Name	Displays the name of the seating section on the drawing, as set in **Section Name**
Display Seating Count	Displays the number of seats in the section on the drawing, separated by table and chair counts
Show Seat Number	Displays a seat number on each symbol; a table-and-chair symbol receives a single number 
Starting Number	Displays and sets the starting number for the first seating symbol in the section
Increment	Specifies the seating number increment value Use an increment value of 2 to number a section with all even or odd seat numbers.
Label Classes	Assign the **Section Name**, **Seating Count**, and **Seat Number** labels to classes to control their appearance in viewports. Select a class present in the drawing or create a new class. When a new class is created, it does not automatically become the active class. Class graphic attributes will be applied to the labels when **Use at Creation** is enabled for the class. (See "Setting Class Properties" on page 179.)
Seating Count	Select a method for limiting the number of seats permitted in a seating area, or leave the option Unlimited. If a limitation method is chosen, specify maximum values for the **Seats per Row** and **Rows per Section**. • Limit the Number of Seats creates no more than the maximum number of seats per row and rows per section specified, fit within the seating area boundaries. • Notify When Count is Exceeded creates seats to fill the seating area but provides red warning text over the seating area when the seats per row and/or rows per section maximum values are exceeded. • Specify the Number of Seats creates exactly the number of seats per row and rows per section specified. All rows are of uniform length; the seating area path is used only to determine an origin and rotation for the seating plan, not to shape or limit the seating area.
Chair Symbol	Displays the seating symbol name used to create the seating layout
Chair Count	Displays the number of seats used in the layout
Table Symbol	Displays the table symbol name used to create the seating layout
Table Count	Displays the number of tables used in the layout
Select Symbol	Opens the Choose a Symbol dialog box, to replace the current seating symbol. Select the desired seating symbol from either the default content or the current file's content, and then click **OK**; see "Resource Libraries" on page 219. When the Spotlight product is installed, the default content for event design seating is available.

Parameter	Description
Symbol Name	Displays the chair or table and chair symbol used to create the seating layout
Vertex Parameters	Edits the seating layout path vertices; see “Editing Vertex-Based Objects” on page 1002.

The seating layout can be reshaped with the **Reshape** tool to add, subtract, and change vertices. The seats are automatically adjusted to fit the new shape. More sophisticated editing operations, such as adding, clipping, intersecting and combining into surfaces can be performed on the seating layout path object by selecting **Modify > Edit Seating Layout**.

The seating section name and seating count text position can be adjusted by moving the control point. Adjust the text font and size of the label and seat numbers with the commands from the **Text** menu.

If the seating orientation requires adjustment, the seat control point can be moved to readjust the seating focal point. The control point is referenced relative to the center of the seating layout. If two imaginary axis lines were placed over the seating layout as shown in the diagram, placing the control point in line with the center of the axes would orient the seats directly toward the control point.



Creating a Seating Layout
Seating Layout Specifications

Seating Layout Specifications

The **Create Seating Layout** command in Vectorworks Architect and Spotlight, as well as the event design suite in the Vectorworks Spotlight product, make use of table and chair symbols that are available from the default content. It’s also possible to make your own table, chair, and nested table and chair symbols for use with these features.

Symbol Characteristics

Symbols should be hybrid (2D/3D) so that they display properly in both 2D and 3D views. At a minimum, the symbol must contain a 2D representation.

Create the 2D view of the symbol using as few polygons and lines as possible. If possible, use a single polyline rather than individual line segments. The line weight of the symbol is also a consideration; the symbols need to stand out when printed. The outer perimeter of the symbol should have a line weight of at least 1/2 point (7 mils). Interior details

should use a lighter line weight. The 2D representation should have a solid fill so that it obscures information under the symbol.

Keep the 3D symbol simple. It should be solid. The model should be accurately sized, but without minute details. These items can add significantly to the rendering time required, and are not necessary to distinguish among instruments.

To create a table and chair symbol, create the chair and table symbols separately, and then create a symbol from the table and chair symbols, so that the table and chair symbols are nested symbols.

Attaching the Event Planning Record to Seating Symbols

Attach the Event Planning Record to the symbol, with the Type field indicating whether the symbol is a table or chair. This allows the Seating Count worksheet to accurately count tables and chairs.

To attach the event planning record:

1. In the Resource Browser, import the Event Planning Record from the Seating.vwx default event planning library file.

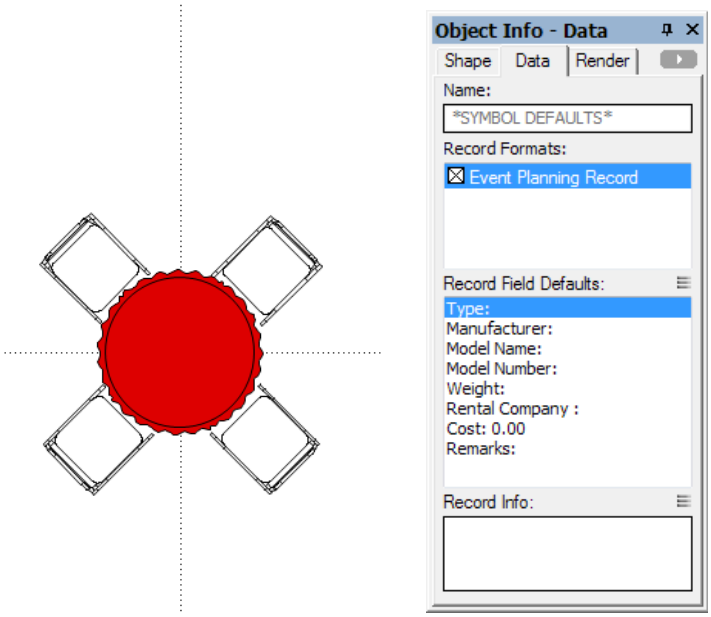
   Locate the Event Planning Record and select **Resources > Import** to bring the record into the current file.

2. Select the new seating symbol, and select **Modify > Edit Symbol**.

   If attaching the record to nested table and chair symbols, select the table or chair, and select **Modify > Edit Symbol** again to edit the individual table or chair. After attaching the event planning record to one of the nested symbols, repeat the procedure to attach the record to the other symbol.

3. In the Edit Symbol window, click on a blank area so that nothing is selected.

4. Click the Data tab in the Object Info palette. Attach the Event Planning Record to the symbol defaults by selecting the check box. Then edit the record by selecting the record field and entering its record information. The **Type** field for tables should contain “Table“ for tables and “Chair” for chairs, for proper object identification and inclusion in worksheets.



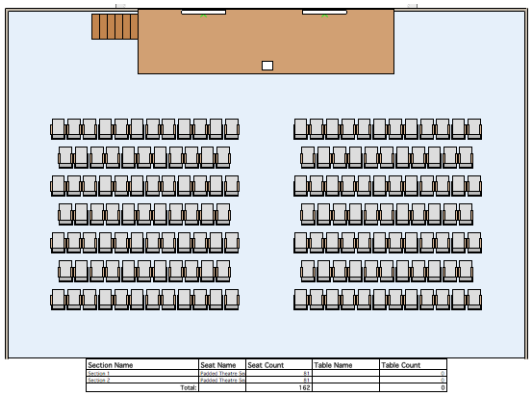
5. Click **Exit Symbol** at the upper right corner of the window to return to the drawing.

Creating Event Views

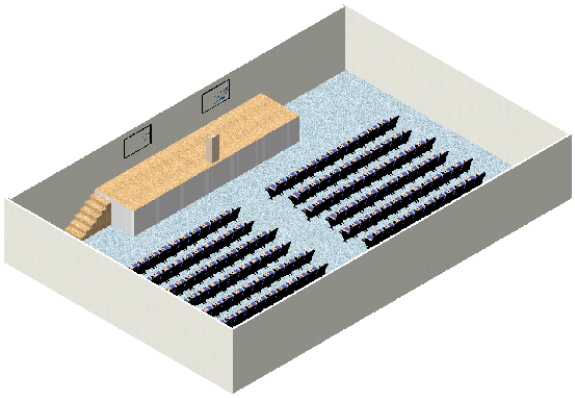
Once the desired elements of the event have been added to the room, it is often necessary to create views of the room to show to the client. The **Create Views** command automatically creates two sheet layers of the event layout: a 2D plan view and a rendered isometric view. The Plan View sheet layer displays the layout in a Top/Plan, Wireframe mode viewport, and includes the seating count worksheet. The Rendered View sheet layer displays the layout in an isometric viewport, rendered in OpenGL mode.

To create a 2D and 3D rendered view of the event layout:

1. Select **Event Design > Create Views**.
2. The views are automatically created. The rendered viewport may require updating; with the viewport selected, click **Update** from the Object Info palette.



Plan view with seating count worksheet



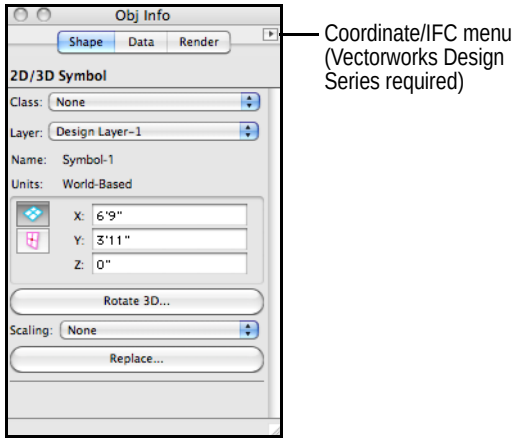
Rendered 3D view; room wall removed for visualization

Using Standard Views
OpenGL
Creating the Room
Creating the Stage
Creating Stage Stairs
Creating a Lectern
Creating a Video Screen
Creating Event Seating

Editing Objects

The Object Info Palette

The Object Info palette provides information about the selected object(s) in a Vectorworks drawing. The information displayed depends on the type of objects selected. In addition, the palette can be used to change object parameters.



Certain objects allow you to right-click (Windows) or Ctrl-click (Mac) on the object to edit and select **Edit** from the context menu to access appropriate editing functions for that object, or select **Properties** from the context menu to access the object's properties dialog box. The object properties dialog box contains information specific to the selected object, similar to the Object Info palette. Double-clicking on an object performs the same function as selecting **Edit** from the context menu (it also reactivates the working plane on which a planar object was created).

The Object Info palette organizes data into three tabbed panes:

Tab	Description
Shape	Displays information about a selected object's geometry, class, layer, and location
Data	Lists any database records attached to a selected object
Render	When the Renderworks product is installed, assigns and maps textures to 3D objects. Also controls mesh smoothing in OpenGL and Renderworks and sketch rendering styles.
Coordinate/ IFC menu (Design Series required)	Specifies options for displaying IFC data (Vectorworks Architect or Landmark required) and for the display of coordinates in rotated plan views (Vectorworks Design Series required). See "Assigning IFC Data to Objects" on page 1746 and "Rotating the Plan" on page 1163.

Object Info palette drop-down list items can be selected by using the mouse or by typing the first letter(s) of the desired item to highlight the closest match in the list (excluding class and layer lists).

Set the Object Info palette position, active pane upon opening, and Data pane separator position, as described in "Saving Initial Palette Positions and Settings" on page 1843. Save palette positions and settings by selecting **Window > Palettes > Save Palette Positions**. To revert the palette position and settings back to the settings established when the custom workspace was created (in the User Data and Preferences folder), click **Reset Saved Settings** from the Session tab of Vectorworks preferences (see "Session Preferences" on page 52).

Activating the Object Info Palette

To activate the Object Info palette with an object selected in the drawing area, select **Window > Palettes > Activate Object Info Palette**, or press the keyboard shortcut to activate the Object Info palette (Ctrl + ‘ (back quote) on Windows or Cmd + Option + C on the Mac).

The keyboard shortcut allows you to quickly switch the focus from the drawing area to the first editable field on the active tab of the Object Info palette. Repeat the keyboard shortcut to switch the focus through the Object Info palette's tabs. If the Object Info palette is not open, or is minimized (unpinned), the Object Info palette is opened, and then the first editable field is highlighted. The shortcut key can be set in the workspace editor.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Shape Tab

Object properties can be directly edited through the Object Info palette from the Shape tab. Objects can also be edited with the tools on the Basic palette as described in the instructions for each tool.

The Shape tab always displays class and layer information. For most objects, **Rotation** displays, indicating the object's orientation. The detailed object information that is also displayed depends on the type of object selected, and can be simple or very extensive depending on the object.

Assigning Objects to Classes and Layers

To assign an object to a class or layer:

1. Select one or more object(s) to assign to the same class and/or layer.
2. Select a class/layer from the list of classes/layers present in the drawing, or create a new class/layer.

   When a new class or layer is created from the Object Info palette, any selected objects are assigned to the new class/layer, but the active class/layer does not change to the newly created class/layer.

   Classes can be displayed in hierarchical order (up to four levels). To enable or disable hierarchical display, see "Session Preferences" on page 52.

Alternatively, objects can be assigned to classes and layers in the Navigation palette (Vectorworks Design Series required).

1. Select one or more object(s) to assign to a class.
2. Select the Classes tab or Design Layers tab from the Navigation palette.
3. Right-click (Windows) or Ctrl-click (Mac) on the class/layer to assign the object(s) to and select **Assign to Selection** from the context menu.

Object Coordinates and Bounding Box Indicator

The Shape tab can display a selected object's plane, coordinate, and bounding box information. The information displayed depends on the type of object, the active plane, and whether it is in a rotated plan view (Vectorworks Design Series required) or positioned relative to a working plane.

For 2D objects, the plane to which the object belongs is indicated by the **Plane** list, and can be changed by selecting another plane from the list. For example, a screen plane object can be turned into a layer plane object by selecting Layer. When a working plane is active, the object can be moved to it. (If a combination of 2D and 3D objects are selected, the **Plane** list selection applies only to the 2D objects.)

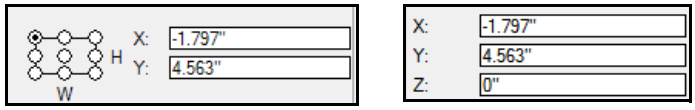
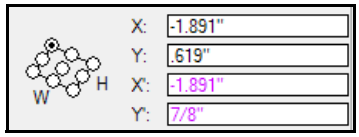
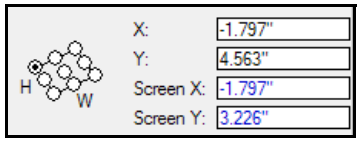
Symbols display the symbol's name and units (whether world- or page-based). Symbols and solids can be scaled from the Object Info palette (see "Scaling Symbols from the Object Info Palette" on page 1064 and "Scaling Solids Asymmetrically" on page 1064.)

The Box Position indicator displays for certain 2D objects such as lines, walls, rectangles, and ovals. It represents the bounding box of the object, indicates which side is the width and height for objects with width and height, and uses a handle to indicate the fixed point about which the object can be resized. The handle is also the location from which the object coordinates are determined. When an object is rotated, the box position indicator rotates also, indicating the
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

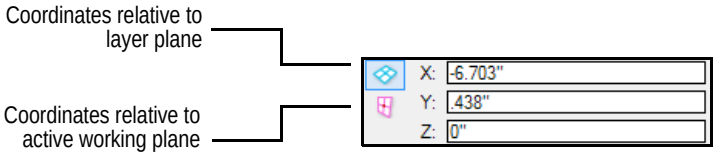
approximate position of the object and, if applicable, the width and height sides. For rectangles, rounded rectangles, and ovals, when the **Rotation** angle is less than ±45° from 90° or 270°, the Vectorworks software automatically swaps the height and width markers for the Box Position indicator and the values in the **Height** and **Width** fields.

Different object coordinates display for an object depending on its plane, position, and plan rotation.

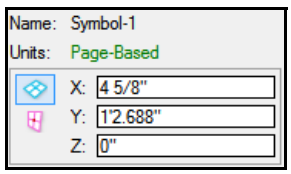
> When a Vectorworks Design Series product is installed, the coordinates menu preference also affects the display of coordinates in rotated plan view (see "Rotating the Plan" on page 1163).

View	Coordinate Display
Top/Plan	Coordinates are relative to the screen plane (X and Y); coordinates for 2D/3D symbols also include the Z coordinate for a streamlined workflow between views 
3D view with working plane active	Coordinates are relative to the layer plane (X and Y) and to the active working plane (X', Y'). The working plane coordinates display in the same color set for the working plane itself in the interactive preferences. 
Rotated plan (Vectorworks Design Series required)	Coordinates are relative to the layer plane (X and Y) and to the rotated plan (Screen X and Screen Y). The rotated plan coordinates display in blue, as do the rulers. 

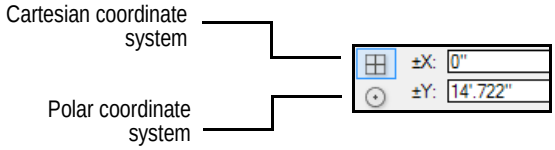
Some 3D objects, such as 3D polygons, NURBS curves, meshes, 3D symbols, spheres, cones, hemispheres, and 3D loci, can display coordinates relative to the layer plane (X, Y, Z)—also called the user coordinate system—or the active working plane (X', Y',Z'), when a working plane is selected. Click the planes button to change the relative coordinate display.



Some symbols are page-based at creation or upon import. These do not display a Box Position indicator in the Object Info palette. (The object can only be resized symmetrically.) The coordinates are relative to the screen plane (X and Y), and 2D/3D symbols also show the Z coordinate for a streamlined workflow between views.



Some objects, such as lines and walls, can display and edit selected object information in either Cartesian or polar mode.



Coordinate System	Description
Cartesian	Cartesian coordinates are the same as those used in the drawing area; values are based on positive and negative X and Y axes
Polar	In polar mode, values are relative to the user origin. Angles are represented as positive or negative values from 0 to 180. Values from 180 to 359 are automatically converted to negative. 0 is at the 3 o'clock position.

Viewing and Editing Single and Multiple Objects

The Shape tab allows either single or multiple objects to be edited.

To view and edit object information for a single object:

1. Select **Window > Palettes > Object Info**.

   The Object Info palette opens.
2. Click the Shape tab.
3. Select the desired object.

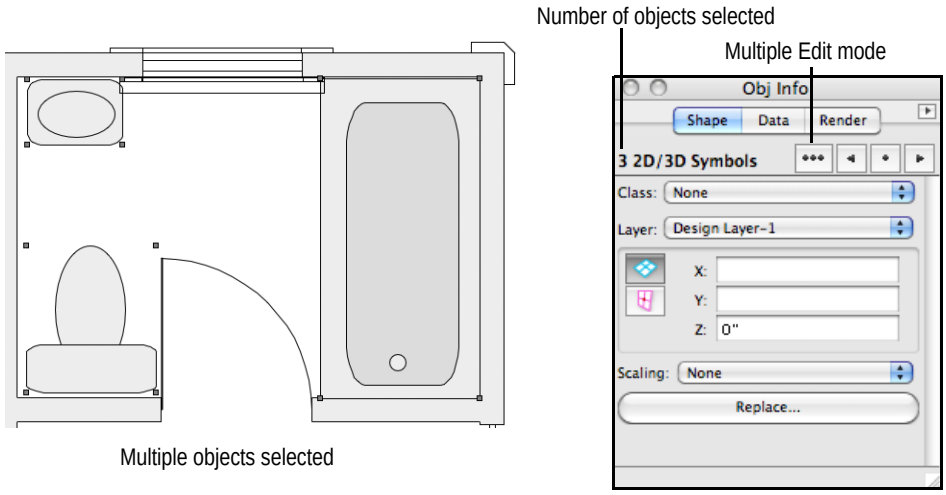
   Editable information for that object is displayed.
4. Change the desired information, using the keys as follows:

Key	Function
Enter	Save the entry and return the focus to the drawing area
Tab	Save the entry and move the focus to the next editable field
Shift+Enter	Save the entry and keep the focus in the same field, so that you can enter a different value if necessary

To view and edit object information for multiple objects:

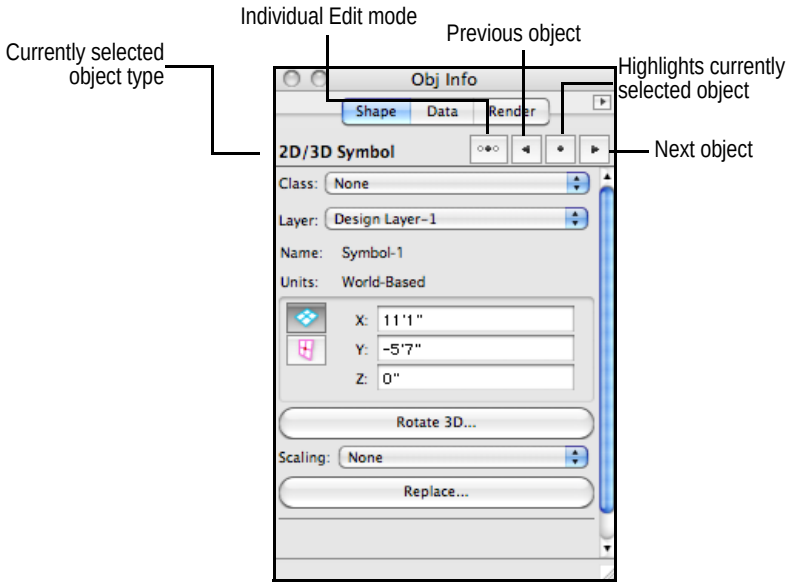
1. Select **Window > Palettes > Object Info**.

   The Object Info palette opens.
2. Click the Shape tab.
3. Select the desired objects.



Multiple objects selected

4. To edit each object in a selection separately, click **Multiple Edit**. This switches to Individual Edit mode. Once in Individual Edit mode, use the Next arrow button to move forward through the selected objects and the Previous arrow button to move backward through the selections. Each object highlights briefly as it is selected. To be sure which object is currently selected, click the center circular button to highlight it again. If either arrow appears dimmed, the end of the selected objects in that order has been reached.

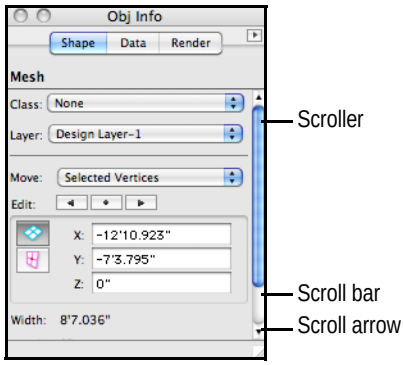


Editing single object within multiple selection

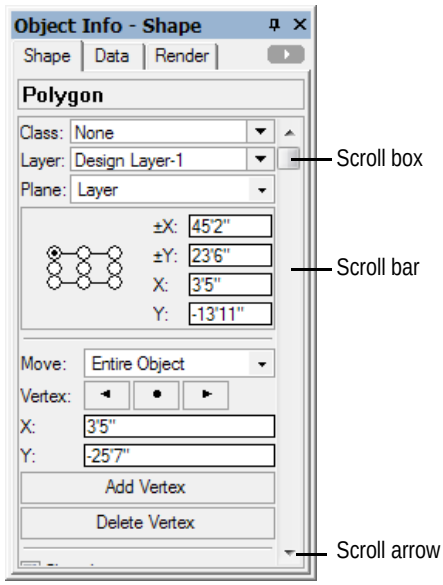
Alternatively, select multiple objects of the same type, and then modify them at one time in Multiple Edit mode. When several objects of the same type have been selected, the palette operates in "batch" editing mode. If the parameter settings of the objects are different, the field displays as blank, or a check box displays with an "indeterminate state." Any parameter edits affect all the selected objects in multiple edit mode.

Some objects have more editable fields than others. If the Object Info palette is not fully extended when an object of this type is selected, scroll options are provided. To scroll up and down, click the scroll arrows, any area of the scroll bar, or click-drag the scroll box (Windows) or scroller (Mac).

Mac

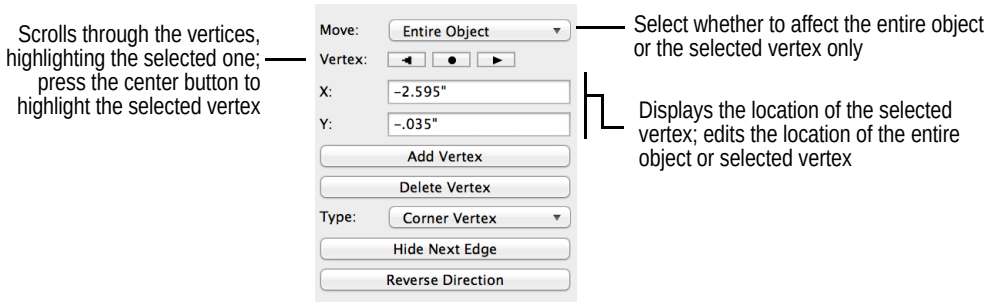


Windows



Editing Vertex-Based Objects

For vertex-based objects, functionality similar to the **Reshape** tool is available from the Object Info palette Shape tab, where values can be entered to move vertices (or the entire object) rather than manipulating them graphically with the tool. Similarly, click the buttons to add, delete, change the vertex type, or hide the next edge; the selected vertex is affected.



Use either the **Selection** tool or the **Reshape** tool from the Basic palette to select a vertex to edit. With the tool selected, Right-click (Windows) or Ctrl-click (Mac) on the vertex (if the SmartCursor is enabled, it snaps to each vertex and identifies the vertex type to assist with selection) and select **Select Vertex in Object Info Palette** from the context menu.

> For certain vertex-based objects, you may first need to select the item and select an edit command to enter object editing mode. A colored border around the drawing window indicates editing mode is active (see “Object Editing Mode” on page 1004).

The X and Y coordinates of the selected vertex display for editing in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

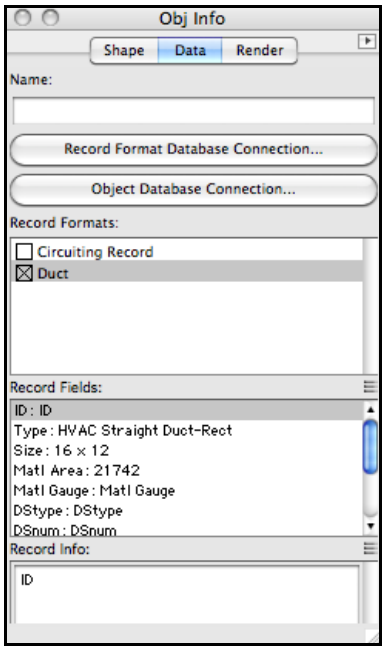
Creating Layers
Creating Classes
The Object Info Palette
Using Arithmetic Expressions
2D Reshape Modes
Data Tab
Render Tab
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Data Tab

The Data tab lists all records in the drawing, along with check boxes showing which records are attached. The Object Info palette can be used to make changes to record settings for individual objects.

To view and edit record information:

1. Select **Window > Palettes > Object Info**.
2. Click the **Data** tab.



ODBC functionality (Vectorworks Design Series required)

3. Select the desired object in the drawing area.

   The following information is displayed.

Click to show/hide the parameters.

Parameter	Description
Name	Displays the name, if any, given to an object
Record Format Database Connection (Deign Series and database connection required)	In the Vectorworks Design Series products, links record formats to an external database for automated, two-way communication.
Object Database Connection (Design Series and database connection required)	Links the object instance to the external database's table rows
Record Formats	Displays all the records that are active in the drawing; if any of these records are attached to the selected object, the box to the left of the record name displays an X. If more than one object is selected, only the records attached to all objects display an X.
Record Fields	Provides a list of all the record fields if a record is attached to the selected object; if any field has a default value assigned, it displays after the field name. If no item is currently selected in the drawing, the label displays **Record Field Defaults** instead of **Record Fields**.

Parameter	Description
Record Info	Edits the field values for the selected record; all entered values override any default values for the object

Resize the list boxes by selecting and dragging the resize bars between them.

See “Record Formats” on page 262 for more information on attaching, editing, and deleting record formats.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Shape Tab
Render Tab
The Object Info Palette
Database Connectivity

Render Tab

When the Renderworks product is installed, this tab is available for the assigning and mapping of textures to 3D objects (see “Applying a Texture to an Object” on page 1530 for a complete description of the parameters available on the Render tab). In two cases, parameters on the Render tab are available in Vectorworks Design Series products, even if Renderworks is not installed: if sketch rendering is active, the **Sketch** parameter is available, and mesh objects rendered in OpenGL can be smoothed using the **Mesh Smoothing** parameter.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Textures and Shaders
Sketch Rendering
Applying Sketch Styles to an Object

Object Editing Mode

Various types of complex objects are edited within a special editing mode in which you edit the components that make up the object. When this editing mode is activated, an Edit window with a colored border displays in the drawing area. Once the edits are complete, exit the object to return to the regular drawing mode.

Depending on the type of object being edited, the Edit window can show the edited object in context with the other objects in the drawing.

- The Edit window shows only the object being edited for the following types of objects: extrudes, multiple extrudes, tapered extrudes, sweeps, meshes, floors, roof faces, and symbol definitions edited from the Resource Browser or from a sheet layer.
- The Edit window can show other objects from the drawing while the following types of objects are being edited: groups, solids, viewports, and symbol definitions that are edited by clicking a symbol instance in a design layer. To enable this feature, select the option to **Show other objects while in editing modes** in the Display tab of the Vectorworks preferences (see “Vectorworks Display Preferences” on page 50). To show the other objects in a less obtrusive way, also select the **Gray other objects** option.

  The **Show other objects while in editing modes** preference does not work when editing a symbol definition from a flipped symbol instance; an alert message displays when this operation is attempted.

To set options for the object editing mode, use the Data Bar and Edit Group Options list on the Tool bar, or select **Window > Edit Group Options**. See “Data Bar and Edit Group Options” on page 127 for details.

To edit an object:

1. Select the object to edit.
2. Select **Modify > Edit** for the object (for example, **Edit Symbol** or **Edit Extrude**).

Alternatively, double-click on the object, or right-click (Windows) or Ctrl-click (Mac) on the object, and then select **Edit** from the context menu.

3. For certain objects, only one component of the object can be edited at a time. When one of these objects is selected, one of the following dialog boxes opens to allow you to select which component to edit. Select a component to edit and click **OK**.

   - Symbol: Edit Symbol dialog box
   - Viewport: Edit Viewport dialog box
   - Path or profile object: Choose Component dialog box

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object, and select a command from the context menu to edit the component directly or, if permitted by the object type, to switch from editing one component to editing another. For example, to edit the path or profile of a plug-in object directly, select **Edit Path** or **Edit Profile** from the context menu.

4. The Edit window displays the item to be edited. A colored border around the drawing window indicates the editing mode is active. The **Exit** command becomes available from the **Modify** menu, and the **Exit** button is visible in the top right corner of the drawing window.

5. Make the changes to the object.

   If other objects in the drawing are displayed, the objects on other layers respect the layer options settings (for showing and snapping to objects in other layers) and the layer visibility settings. Other objects on the same layer as the edited object are snappable. One exception to this is when editing page-based (green) symbols, which are edited at a 1:1 scale; other objects which are not at a 1:1 scale are not snappable.

   In addition, the view can be switched between regular and unified view mode as needed. (See “Unified Layer View” on page 1152 for more information.)

6. Click the **Exit** button to return to the regular drawing mode. (If a nested object is being edited, the **Exit** button exits back to the next level.)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing a Group
Editing Symbol Definitions

Moving Objects

Objects can be moved in several ways.

- Use the **Move** command (operates in the screen plane) or **Move 3D** command to move an object an exact distance.
- Use the **Move by Points** tool to move, duplicate, and distribute objects by clicking.
- Use the selection tools to select and drag objects to a new location.
- Nudge selected objects one pixel at a time, or by the snap grid (see “Edit Preferences” on page 49).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Objects in the Screen Plane
Moving Objects in 3D Space
Moving Objects by Clicking with the Move by Points Tool
Moving Symbols in Walls with the Selection Tool
Nudging Symbols in Walls
Moving Symbols in Walls with the Move Command
Moving Symbols in Walls with the Move by Points Tool

Moving Objects in the Screen Plane

For information about moving symbols in walls, see “Moving Symbols in Walls with the Move Command” on page 257.

To move an object in screen plane space with the **Move** command:

1. Select the object(s) to move.
2. Select **Modify > Move > Move**.

   The Move Selection dialog box opens. Move the objects according to Cartesian or polar coordinates.

Click to show/hide the parameters.

Parameter	Description
Cartesian	Specify the distance to move the object; for Cartesian coordinates, select **Cartesian** and enter the **X** and **Y Offset** distances
Polar	Specify the distance to move the object; for polar coordinates, select **Polar** and enter the **Distance** and **Angle**

3. Click **OK**.

   The object is moved in the screen plane according to the criteria specified.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Objects

Moving Objects in 3D Space

To move an object with the **Move 3D** command:

1. Select the object(s) to move.
2. Select **Modify > Move > Move 3D**.

   The Move 3D Selection dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Cartesian	Select the X, Y, Z (Cartesian) coordinate system
X, Y, Z Offset	For Cartesian coordinates, specify the distance to move the object
Working Plane	Select the X’, Y’, Z’ (working plane) coordinate system
X’, Y’, Z’ Offset	For working plane coordinates, specify the distance to move the object

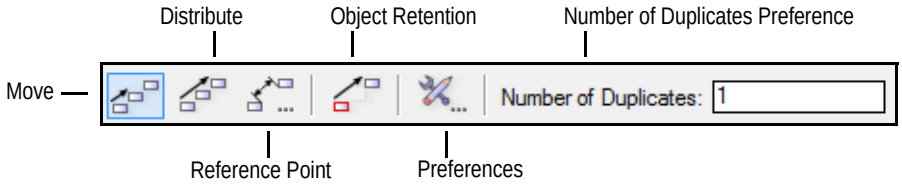
3. Click **OK**. The object is moved in 3D space according to the criteria specified.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Objects

Moving Objects by Clicking with the Move by Points Tool

Objects can be moved, duplicated, and distributed along a specified distance by clicking with the **Move by Points** tool. This tool also moves symbols within walls (see “Moving Symbols in Walls with the Move by Points Tool” on page 258).



Mode	Description
Move	Moves and duplicates selected objects according to the distance and direction specified by two mouse clicks
Distribute	Moves and distributes duplicate objects between the points specified by two mouse clicks
Reference Point	Moves selected objects according to a clicked reference point and a specified offset distance from that point; the Preferences settings are not applicable when this mode is selected
Object Retention	Keeps the original objects; this is the same as selecting the **Retain** option in the tool preferences

Click here for a video tip on this topic (Internet access required).

Moving Objects with the Move or Distribute Mode

To move, duplicate, and distribute selected objects:

1. Select the object(s) to move and/or duplicate.
2. Click the **Move by Points** tool from the Basic palette and click **Preferences** from the Tool bar.

   The Move by Points Settings dialog box opens. Select the settings and click **OK**.

Click to show/hide the parameters.

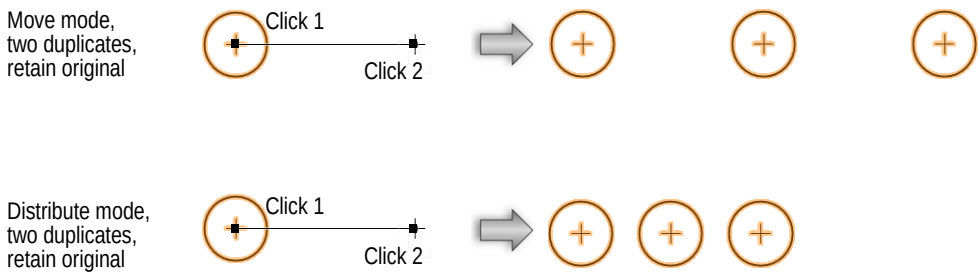
Parameter	Description
Number of Duplicates	Specifies the number of copies of the original object(s) to create; same as the **Number of Duplicates** field on the Tool bar. A value of 0 (zero) means that the original object will be moved; in this case, the **Retain** option below (and the **Object Retention** button on the Tool bar) have no effect.
Original Object	
Retain	Keeps the original object(s); this option performs the same function as the **Object Retention** button on the Tool bar
Leave Selected	Keeps the original object(s) selected for further action

To select additional objects or to change the currently selected objects to move, duplicate, or distribute, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd key is being pressed.

3. Select either the Move or Distribute mode, depending on the desired outcome.

Mode	Number of Duplicates	Original Object Retained	Result
Move or Distribute	0	not applicable	Moves the original object(s) to the location specified by clicking
Move or Distribute	1	No	Moves the original object(s) to the location specified by clicking
Move or Distribute	1	Yes	Creates one duplicate of the original object(s), placed at the location indicated by clicking
Move	> 1	No	Moves the original object(s), placing the original at the location specified by the first click, and placing the duplicates the same distance apart from each other along the vector created by the two click points
Move	> 1	Yes	Creates duplicates of the original object(s), placing the first duplicate at the location specified by the second click, and placing additional duplicates the same distance apart from each other along the vector created by the two click points
Distribute	> 1	No	Moves the original object(s), placing the original at the location specified by the first click, and distributing the duplicates between the two click points, and along the vector created by the two points
Distribute	> 1	Yes	Creates duplicates of the original object(s), distributing the duplicates between the two click points, and along the vector created by the two points

4. Click once (anywhere on the drawing) to indicate the vector start point, and click again to indicate the vector end point. Duplicates are placed in the vector direction, at the distance specified by the click points, along the line defined by the click points.



Moving Objects with the Reference Point Mode

Reference Point mode can work two ways; the first click can either indicate the object to be moved or the reference point from which the object will be offset.

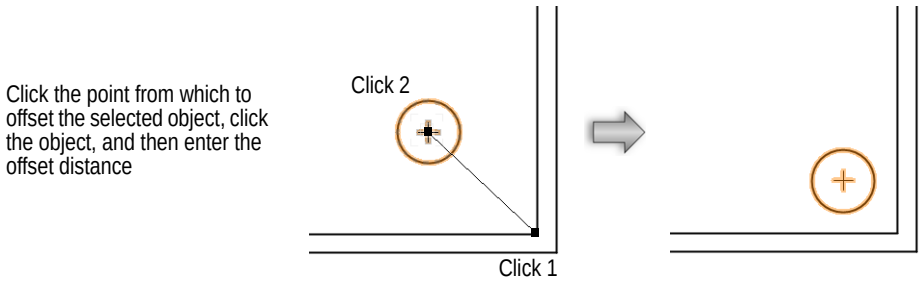
To move selected objects using a reference point:

1. Select the object(s) to move.
2. Click the **Move by Points** tool from the Basic palette.

   To select additional objects or to change the currently selected objects to move, duplicate, or distribute, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd is being pressed.

3. Select the **Reference Point** mode from the Tool bar.
4. Click to indicate either the reference point, or a point on the selected object.

   The Enter Offset dialog box opens.
5. The current **Offset** distance between the two click points displays. Enter the desired distance.
6. Select whether the first click made was the reference point or a point on the object to be moved.
7. Click **OK**. The selected object is moved as specified, along the line defined by the click points.
8. The setting for how the first click is interpreted remains set until it is changed.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving Objects

Cutting, Copying, and Pasting Objects

Cutting Objects

The **Cut** command removes an object from the drawing, temporarily storing the object in the clipboard.

To cut an object:

1. Select the object(s) to remove.
2. Select **Edit > Cut**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object, and select **Cut** from the object context menu.

   The object is moved from the drawing to the clipboard.

Copying Objects

The **Copy** command copies an object to the clipboard, where it is temporarily stored. The original object remains on the drawing.

To copy an object:

1. Select the object(s) to copy.
2. Select **Edit > Copy**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object, and select **Copy** from the object context menu.

   The program places a copy of the object in the clipboard.

Pasting Objects

The **Paste** command places the clipboard object(s) into the same drawing file, into another Vectorworks file, or into another software program's file (if that program also has copy, cut, and paste commands). As long as the Vectorworks
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

program remains open while the object is in the clipboard, the object retains all its object information for pasting into Vectorworks documents.

Some image quality can be lost when Vectorworks objects are pasted into other programs, or from other programs into a Vectorworks file.

To paste an object:

1. Open the file and layer where the object is to be added.
2. Select **Edit > Paste**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the drawing area, and select **Paste** from the document context menu.

   The program pastes the clipboard contents centered on the location of the last mouse click, unless the last mouse click is outside the current view. In that case, the contents are pasted at the center of the drawing. When you paste into a different Vectorworks file, the program automatically centers the clipboard contents in the drawing area.

   When a raster image is pasted into a Vectorworks file, it is automatically compressed to PNG format to reduce the Vectorworks file size.

Paste as Bitmap

Like the **Paste** command, the **Paste as Bitmap** command places a copy of the clipboard contents in the active Vectorworks drawing file. However, **Paste as Bitmap** places the entire clipboard contents as a single bitmap object. Because the pasted clipboard content is now one item, individual objects (including symbols and text) can no longer be edited.

This command can be useful if there are multiple objects on the clipboard that you do not want to edit separately, or if the clipboard objects may lose some image quality if they are converted into Vectorworks objects.

To paste as a bitmap:

1. Open the file and layer where the object is to be added.
2. Select **Edit > Paste as Bitmap**.

Paste in Place

The **Paste in Place** command works exactly like the **Paste** command, except that the clipboard contents are pasted into the active drawing at the same coordinates (relative to the user origin) from which they were copied. When you paste to a sheet layer, remember that each sheet layer has its own user origin, which may be different from the origin of the layer where the object was copied (see “Internal Origin and User Origin” on page 72 for details).

To paste in place:

1. Open the file and layer where the object is to be added.
2. Select **Edit > Paste in Place**.

Changing Object Stacking Order

As objects are drawn, the program keeps track of their stacking order within the design layer. The first object created is at the back of the stack, and the most recent object created is at the front of the stack.

The send commands change the stacking order of objects within a layer. Objects can be sent forward to be in front of an overlapping object or sent backward to be behind an overlapping object. Also, objects can be sent all the way to the front or back of the stack in one step.

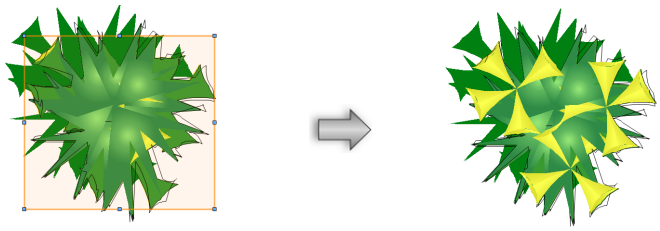
To change the stacking order of an object:

1. Select the object to restack.
2. Select **Modify > Send**. Select the send method to use.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object, and select **Send** from the object context menu.

Method	Description
Send to Front	Sends the selected object to the front of stack
Send Forward	Sends the selected object toward the front of the stack, in front of the next object in the stack that overlaps it; if no other object overlaps the selected object, the stacking order is not changed
Send to Back	Sends the selected object to the back of the stack
Send Backward	Sends the selected object toward the back of the stack, behind the next object in the stack that overlaps it; if no other object overlaps the selected object, the stacking order is not changed

The object's stacking order is changed.



Removing Objects

Clearing Objects

The **Clear** command deletes any selected object or objects. It has the same effect as pressing the Delete key, meaning that the object(s) are not stored on the clipboard. The only way to retrieve a "cleared" object is to select **Undo**. This command provides an additional way of removing an object from the drawing without deleting the current contents of the clipboard.

To clear an object from the drawing:

1. Select the object or objects to remove from the drawing.
2. Select **Edit > Clear**.

Purging Items from a File

Purging Items from a File

The **Purge** command removes specified items from the drawing and reduces the file size. Unused layers, classes, and resources, as well as objects outside of the page boundaries can be removed from the file.

To purge items from a file:

1. Select **Tools > Purge**.

   The Purge dialog box opens. Each type of item that can be purged is listed, along with the number of items that were found. Select the items to remove from the drawing file.

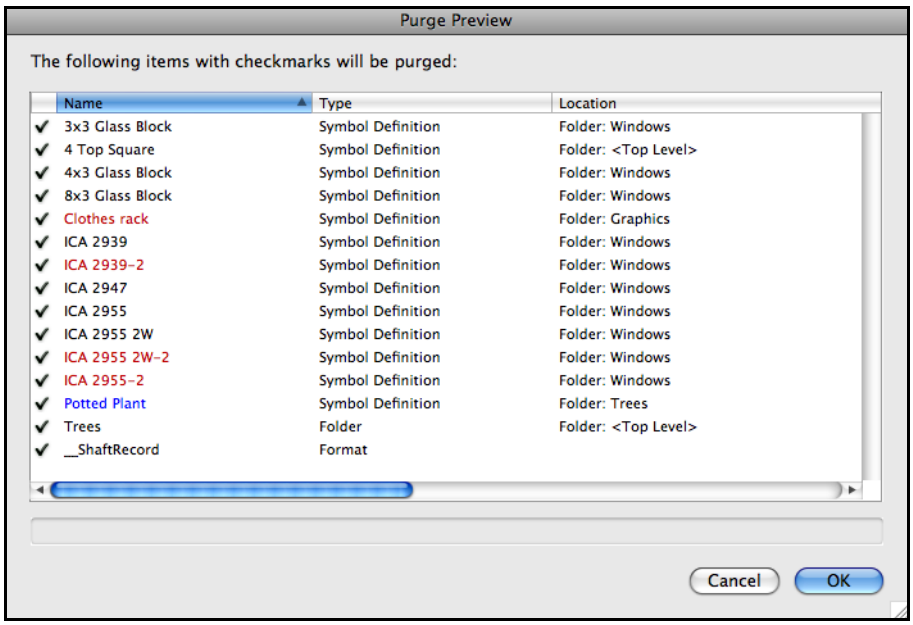
Item	Description
Unused Resources	
Gradients	Deletes any unused gradient resources
Hatches	Deletes any unused hatch resource. If a surface hatch's associated texture is in use (Renderworks required), that hatch is also considered in use and is not purged.
Images	Deletes any unused image resources
Line Types	Deletes any unused line type resources
Record Formats	Deletes any unused record format resources. Also select **Special Record Formats** to delete unused record formats that store plug-in object default values. An alert prompts you to confirm that the default values should be deleted.
Renderworks Backgrounds (Renderworks required)	Deletes any unused Renderworks background resources
Renderworks Styles (Renderworks required)	Deletes any unused Renderworks render style resources
Renderworks Textures (Renderworks required)	Deletes any unused texture resources
Sketch Styles (Vectorworks Design Series required)	Deletes any unused sketch style resources
Slab Styles (Vectorworks Architect required)	Deletes any unused slab style resources
Symbols	Deletes any unused symbol resources. Also select **Special Symbols** to delete the resources for plug-in object symbols and group symbols (those listed in red and blue in the Resource Browser). This will not delete any associated plug-in or group objects that have already been placed in the drawing. An alert prompts you to confirm that the special symbol resources should be deleted.
Text Styles	Deletes any unused text style resources
Tiles	Deletes any unused tile resources
Wall Styles (Vectorworks Architect or Landmark required)	Deletes any unused wall style resources

Item	Description
All	• **Checked box:** All attributes in the Unused Resources group are selected; click to deselect all attributes in the group. • **Empty box:** No attributes in the Unused Resources group are selected; click to select all attributes in the group. • **Box displays (Windows) or (Mac):** One or more attributes in the Unused Resources group are not selected; click to select all attributes in the group. The **Special Record Formats** and **Special Symbols** options are not automatically selected along with the other unused resources; they must be selected individually.
Other Items	
Empty Layers	Deletes any layers that do not contain any objects
Empty Story Layers (Vectorworks Architect required)	Deletes any stories that do not contain any layers
Empty Symbol Folders	Deletes any unused symbol folders
Unused Classes	Deletes any unused classes
Objects Outside of Page Boundaries for	Deletes any objects that are totally outside the page boundaries; if an object is partially within the boundary, it will not be deleted. Objects can be deleted from the active layer only, from all design layers, and from all sheet layers.
All	• **Checked box:** All attributes in the Other Items group are selected; click to deselect all attributes in the group. • **Empty box:** No attributes in the Other Items group are selected; click to select all attributes in the group. • **Box displays (Windows) or (Mac):** One or more attributes in the Other Items group are not selected; click to select all attributes in the group.
Preview items that will be purged	Opens the Purge Preview dialog box after you click **OK** on the Purge dialog box; if needed, items can be removed from the purge list

2. Click **OK**.

   If **Preview items that will be purged** was selected, the Purge Preview dialog box opens. The Name and Type of each item that will be purged displays. Red and blue names indicate symbols that are defined to be inserted as plug-in objects and groups, respectively (see “Symbol Types” on page 237). For symbol definitions and empty symbol folders, the Location column indicates which resource folder currently contains the item. For objects that are outside of the page boundaries, the Location column indicates which sheet layer or design layer currently contains the item.

   To prevent an item from being purged, click the column to the left of the item’s name to remove the check mark.



3. Click **OK** to complete the purge.
4. If necessary, use the **Undo** command to undo the purge.

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Clearing Objects

Duplicating Objects

Duplicating Single

The **Duplicate** command makes a copy of an object or group of objects and places it on the drawing. Depending on the setting in Vectorworks preferences, the duplicated object or objects will either be offset or placed directly on top of the original.

To duplicate any object:

1. Select the object(s) to copy.
2. Select **Edit > Duplicate**.

   The program places a copy of the selected object(s) in the drawing according to the settings in the Vectorworks preferences dialog box.

   When objects with an offset are duplicated, the offset is maintained with the duplicate.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Duplicate Array
Duplicating Objects Along a Path

Duplicate Array

The **Duplicate Array** command controls how many copies of selected objects are made and how these copies are arrayed, or placed, in the drawing.

For information about placing symbols in walls in a duplicate array, see “Inserting Symbols in a Duplicate Array” on page 255.

To create a duplicate array:

1. Select the object or objects to copy.
2. Select **Edit > Duplicate Array**.

   The Duplicate Array dialog box opens. Select the desired duplication array **Shape**. The dialog box dynamically displays the appropriate fields based on the selected linear, rectangular, or circular array shape.

Linear Array Rectangular Array Circular Array

Linear Array

Click to show/hide the parameters.

Parameter	Description
Number of Duplicates	Specify the number of copies of the original object to create
First Duplicate's Position Determined By	Specify the start point for the first copied object, by either specifying coordinates or placing the object with the mouse
Cartesian Offset	To use Cartesian coordinates, click this option and specify the distance of the center of the copy from the center of the original object by entering **X**, **Y**, and **Z** coordinates. If the view is something other than Top/Plan, and the field **Values Are With Respect to the** is set to Working Plane, the labels on the coordinate fields are **X'**, **Y'**, and **Z'** instead of **X**, **Y**, and **Z**. **Z** (or **Z'**) must be zero when only 2D screen objects are being duplicated.
Polar Offset	This option is always enabled if 2D objects are selected. If 3D or hybrid objects are selected, or if both 2D and 3D objects are selected, this option is enabled only in Top/Plan view. To use polar coordinates, click this option and specify the distance of the center of the copy from the center of the original object by entering **r** (radius), theta (angle), and **Z** coordinates; **Z** must be zero when only 2D screen objects are being duplicated.
Next Mouse Click and Z (or Z')	This option is always enabled if 2D objects are selected. If 3D or hybrid objects are selected, or if both 2D and 3D objects are selected, this option is enabled only in Top/Plan view. To place the copy with the mouse, select **Next Mouse Click**. To offset the first copy from the original object's plane, enter a **Z** (or **Z'**) value; **Z** (or **Z'**) must be zero when only 2D screen objects are being duplicated.
Values Are With Respect to the	Specify whether the array should be built relative to the active layer plane or working plane
Resize Duplicates	Select **Resize Duplicates** to resize each successive copy with the specified **X Scale**, **Y Scale**, and **Z Scale** values; to leave the copies at the same scale as the original object, ensure that **Resize Duplicates** is deselected
Rotate Duplicates	Select **Rotate Duplicates** to rotate each successive copy at the specified **Angle**; to leave the copies at the same rotation as the original object, ensure that **Rotate Duplicates** is deselected
Original Object	To include the original object in the array, select **Retain**; otherwise, the original object is deleted. To leave the original object selected after duplication, also click **Leave Selected**.

Rectangular Array

Click to show/hide the parameters.

Parameter	Description
Number of Columns	Specify the number of copies of the original object to create in the array's X direction (active layer plane duplication) or X' direction (working plane duplication)
Number of Rows	Specify the number of copies of the original object to create in the array's Y direction (active layer plane duplication) or Y' direction (working plane duplication)
Number of Stacks	Specify the number of copies of the original object to create in the array's Z direction (active layer plane duplication) or Z' direction (working plane duplication)
Distance Between Columns	Specify the distance between each column in the array (measured from the copied objects' centers)
Distance Between Rows	Specify the distance between each row in the array (measured from the copied objects' centers)
Distance Between Stacks	Specify the distance between each stack in the array (measured from the copied objects' centers); this value must be zero when only 2D screen objects are being duplicated
Values Are With Respect to the	Specify whether the array should be built relative to the active layer plane or working plane
Resize Duplicates	Select **Resize Duplicates** to resize each successive copy with the specified **X Scale**, **Y Scale**, and **Z Scale** values; to leave the copies at the same scale as the original object ensure that **Resize Duplicates** is deselected
Rotate Duplicates	Select **Rotate Duplicates** to rotate each successive copy at the specified **Angle**; to leave the copies at the same rotation as the original object, ensure that **Rotate Duplicates** is deselected
Original Object	To include the original object in the array, select **Retain**; otherwise, the original object is deleted. To leave the original object selected after duplication, also click **Leave Selected**.

Circular Array

Click to show/hide the parameters.

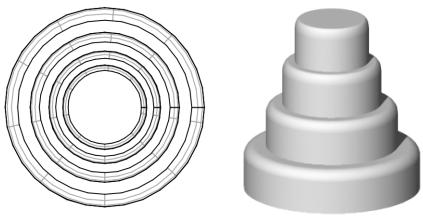
Parameter	Description
Number of Duplicates	Specify the number of copies of the original object to create
Angle Between Duplicates	Specify the angle of separation between the centers of each object in the array
Z (or Z') Offset of Successive Duplicates	To offset copies from the original object's plane, specify the distance between each successive copy. If the view is something other than Top/Plan, and **Values Are With Respect to the** is set to Working Plane, the label is **Z'** instead of **Z**. **Z** and **Z'** must be zero when only 2D screen objects are being duplicated.
Circle Center Point	Specify the center point for the circular array, by either specifying coordinates or placing the array with the mouse

Parameter	Description
X and Y (or X' and Y')	To place the array using coordinates, enter the **X** and **Y** coordinates of its center point. If the view is something other than Top/Plan, and **Values Are With Respect to the** is set to Working Plane, the labels on the coordinate fields are **X'** and **Y'** instead of **X** and **Y**.
Next Mouse Click	This option is always enabled if 2D objects are selected; if 3D or hybrid objects are selected, or if both 2D and 3D objects are selected, this option is enabled only in Top/Plan view. Select **Next Mouse Click** to place the array so that its center is at the next clicked point.
Values Are With Respect to the	Specify whether the array should be built relative to the active layer plane or working pane
Resize Duplicates	Select **Resize Duplicates** to resize each successive copy with the specified **X Scale**, **Y Scale**, and **Z Scale** values; to leave the copies at the same scale as the original object, ensure that **Resize Duplicates** is deselected
Rotate Duplicates	Select **Rotate Duplicates** to rotate each successive copy at either the **Angle Between Duplicates** or the specified **Custom Angle**; to leave the copies at the same rotation as the original object, ensure that **Rotate Duplicates** is deselected
Original Object	To include the original object in the array, select **Retain**; otherwise, the original object is deleted. To leave the original object selected after duplication, also click **Leave Selected**.

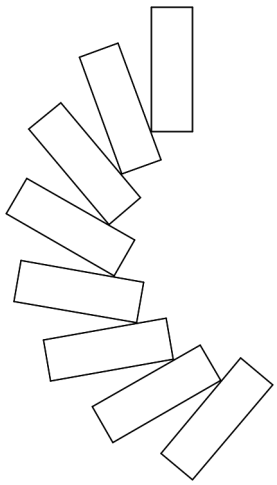
3. Set the parameters and click **OK**.

   If the array location is already specified, the program automatically places the duplicate array.

   If **Next Mouse Click** was selected, move the mouse where the copies are to be placed, and click. For circular and rectangular arrays, click the mouse at the center of the array.



3D linear array with Z offset and resized duplicates



3D circular array with Z offset and rotated duplicates

Duplicating Single
Duplicating Objects Along a Path

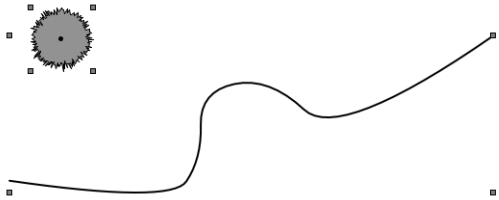
Duplicating Objects Along a Path

The **Duplicate Along Path** command creates and places several copies of an object or objects along an existing path.

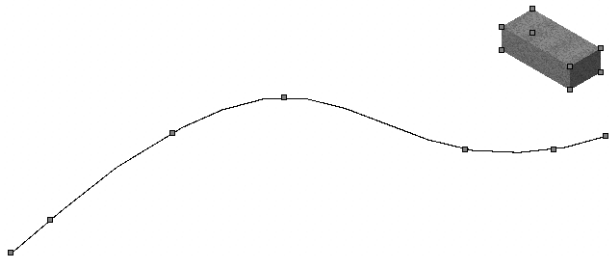
If a 2D object and 3D path are selected, the 2D object is projected onto the path.

To duplicate objects along a path:

1. Select the object or objects to duplicate, and select the path object.



Grouped 2D objects and polyline path

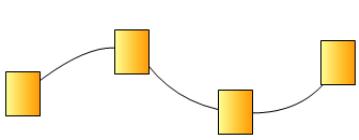
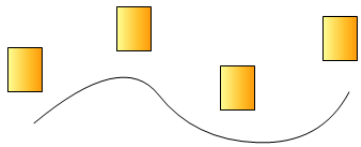
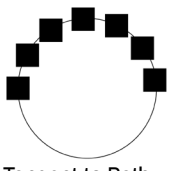
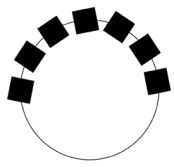


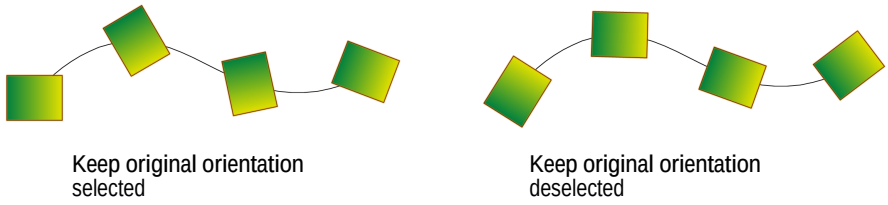
3D object (in 3D view) and NURBS curve

2. Select **Edit > Duplicate Along Path**.

   The Duplicate Along Path dialog box opens.

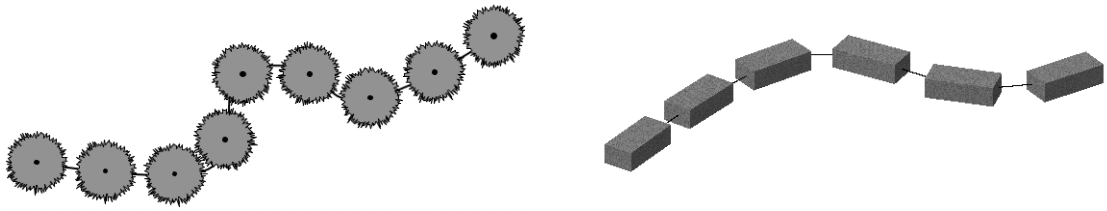
Click to show/hide the parameters.

Parameter	Description
Select a path object	Click **Prev** or **Next** to select the object that represents the path; the selected path object is highlighted
Duplicate Placement	Specifies the duplicate object placement parameters
Number of Duplicates	Creates the specified **Number** of duplicate objects, equally spaced along the path
Fixed Distance	Duplicates objects at the fixed intervals specified in **Distance**
Start Offset	Specifies the distance from the end of the path to the first duplicated object; enter zero to place the first object at the start of the path
Curve Length	Displays the length of the path object, for reference
Center object on path	Centers original and duplicates along the path; deselect the option to maintain the position of the duplicates relative to the path  Center object on path selected  Center object on path deselected
Tangent to path	Rotates the duplicates so they are always tangent to the path  Tangent to Path deselected  Tangent to Path selected

Parameter	Description
Keep original orientation	Maintains the original object's position and rotates the duplicates tangent to the path, relative to the original object's tangency angle  Keep original orientation selected Keep original orientation deselected
Preview	Click to preview the effect of parameter changes before clicking **OK**

3. Click **OK**.

   The duplicates are arrayed with their centers along the selected path object.



Duplicating Single
Duplicate Array

Smoothing Objects

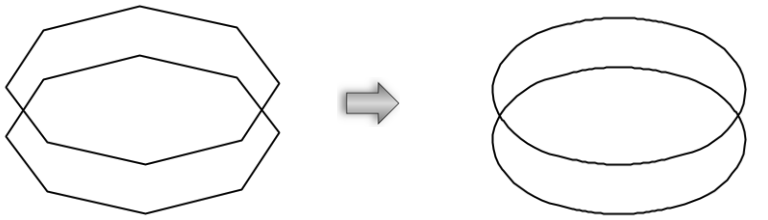
The smoothing commands smooth all selected polygon and polyline vertices in one step, changing all vertices of the object into the selected vertex type. The commands also work on NURBS curves. The **No Smoothing** command changes smoothed vertices into cornered vertices.

To smooth an object:

1. Select the object(s) to smooth.
2. Select **Modify > Poly Smoothing**. Select the smoothing method to use.

Method	Description
No Smoothing	Creates straight lines and corner vertices
Bézier Spline Smoothing	Creates curves pulled towards but not touching the vertices
Cubic Spline Smoothing	Creates curves that pass through the control points
Arc Smoothing	Creates fillet-like curves at the control points. The arc radius is based on the current fillet radius of the **Fillet** tool (see "Fillet Tool" on page 1077). If no radius is set, the largest radius that can fit between each vertex is used.
Radius Smoothing	Creates curved radius vertices, which are easier to reshape than arc vertices

   The object is smoothed.



Smoothed using Arc Smoothing

Simplifying Polygons and Polylines

Composing and Decomposing Objects and Surfaces

Composing Objects and Surfaces

The **Compose** command can combine NURBS surfaces, lines, arcs, open polygons, NURBS curves, NURBS arcs, and open 3D polygons into a single object. The endpoints of each object must be touching to use this command. Objects that are not touching are ignored.

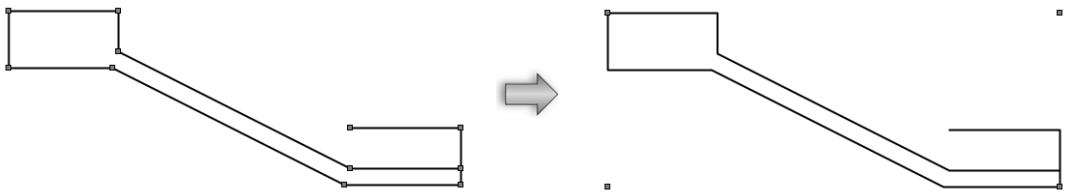
Adjacent NURBS surfaces, which may have been split by the **Split** tool (see "Splitting Objects and NURBS Surfaces with the Split Tool" on page 1072), can be composed into a single NURBS surface with this command.

To compose adjacent objects or NURBS surfaces:

1. Select the objects or NURBS surfaces to be composed.

   If a large number of objects are connected end to end, use the **Edit > Select Connected Objects** command to select them.

2. Select **Modify > Compose**. The objects are combined to create a single object or NURBS surface.



If there are multiple objects touching at the same endpoint, the two objects closest in stacking order are combined.

Selecting Connected Objects
Decomposing Objects and Surfaces

Decomposing Objects and Surfaces

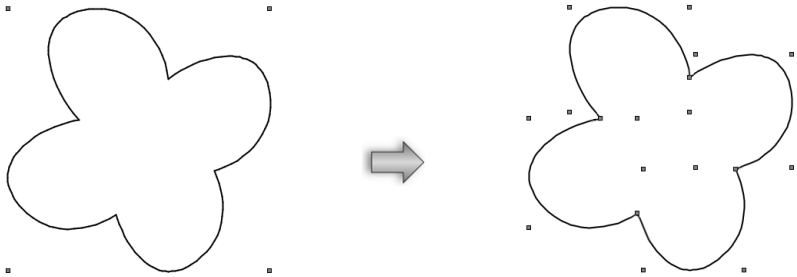
The **Decompose** command can decompose NURBS surfaces, 2D polygons, 2D polylines, 3D NURBS curves created with the **Compose** command, and open 3D polygons.

Occasionally, surface creation methods like the **Create Surface from Curves** command can create a surface with internal discontinuities. Many of the 3D Power Pack tools and commands are not designed to work with these types of surfaces. The **Decompose** command can separate the surface into NURBS surfaces without discontinuities.

To decompose a NURBS surface or object into individual segments:

1. Select the NURBS surface or object to be decomposed.
2. Select **Modify > Decompose**.

The NURBS surface is separated into individual surfaces, or the object is separated into individual segments.



Composing Objects and Surfaces
Selecting Connected Objects

Selecting Connected Objects

The **Select Connected Objects** command can select objects that are connected end to end with one or more selected objects. Any combination of connected lines, arcs, open polygons, open polylines, open NURBS curves, open 3D polygons, walls, and open round walls can be selected. The endpoints of each object must be touching either directly or indirectly to use this command. Objects that are not touching are ignored.

This command can be useful when an imported DXF/DWG file contains a large number of small lines that are connected end to end and you want to compose them into a single object.

To select connected objects:

1. Select one or more objects to which the other objects to be selected are connected end to end.
2. Select **Edit > Select Connected Objects.**

   The selected objects are highlighted; the Object Info palette indicates how many objects were selected.

Composing Objects and Surfaces

Locking and Unlocking Objects

Locking Objects

Objects in a drawing can be protected with the **Lock** command so that they cannot be accidentally moved, deleted, or edited. A locked object must be unlocked before any changes can be made to it.

To lock an object:

1. Select the object or objects to lock.
2. Select **Modify > Lock.**

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object, and select **Lock** from the object context menu.

The Object Info palette indicates that the object is locked. Depending on the selection highlighting setting in Vectorworks preferences, the handles and/or highlighting also change to indicate that the object is now locked (see “Selection and Pre-selection Indicators” on page 114).

Unlocking Objects

Unlock an object or group of objects that was previously locked with the **Unlock** command. Unlocked objects can be copied, moved, deleted, or edited.

To unlock objects:

1. Select the object or objects to be unlocked.
2. Select **Modify > Unlock**

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object, and select **Unlock** from the object context menu.

   The Object Info palette no longer indicates that the object is locked. Depending on the selection highlighting setting in Vectorworks preferences, the handles and/or highlighting also change to indicate that the object is now unlocked (see “Selection and Pre-selection Indicators” on page 114).

Rotating Objects

There are several ways to rotate objects. Use the **Rotate** tool to rotate an object, or a duplicate of an object, directly with the mouse. Use the various **Rotate** commands to rotate the object by choosing a preset rotation, or by entering custom rotation information in a dialog box.

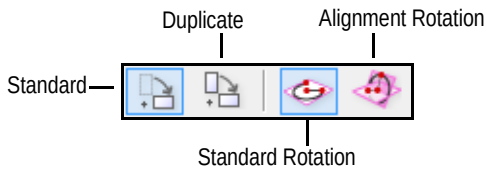
Certain objects, such as rectangles, rounded rectangles, ovals, bitmaps, and text objects, can be rotated with the Interactive Scaling mode of the **Selection** tool. Press the Alt key (Windows) or Option key (Mac) while dragging a reshape handle.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Rotate Tool

The **Rotate** tool can rotate objects in the drawing. Double-clicking the tool when an object is selected opens the Rotate Object dialog box as described in “Custom Rotation” on page 1025.

The **Rotate** tool can rotate, or rotate and duplicate the selected object(s) about an axis or aligned relative to another object.



Mode	Description
Standard	Rotates the selected object
Duplicate	Creates a duplicate object and rotates it
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mode	Description
Standard Rotation (3D views only)	Rotates the object about a defined axis
Alignment Rotation (3D views only)	Rotates the object by aligning it with another object

Click here for a video tip on this topic (Internet connection required).

Standard Rotation
Rotation by Alignment
Rotating Objects

Standard Rotation

To rotate an object around a specified axis:

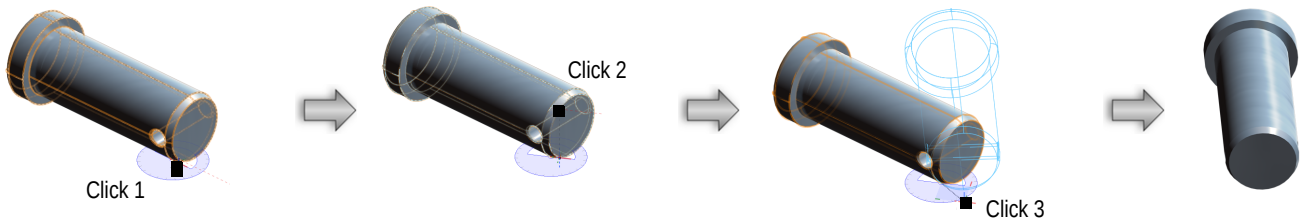
1. Select the object to rotate.
2. Click the **Rotate** tool from the Basic palette.

   To select additional objects or to change the currently selected objects to rotate, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd key is being pressed.
3. Select **Standard** or **Duplicate** from the Tool bar, depending on whether the original or a duplicate object is to be rotated.
4. In a 3D view, select **Standard Rotation** from the Tool bar.

   By default, the rotation plane is that of the active working plane; the working plane orientation can be changed. If desired, select **Screen Plane** from the Active Plane list on the Tool bar to rotate in a plane parallel to the screen plane.
5. A protractor feedback graphic displays around the cursor. Position the protractor on the appropriate rotation plane, and click at the center point of the rotation.

   The protractor graphic displays only when an object is selected.
6. The protractor graphic remains at the click point. As you move the cursor, the protractor rotates, and a dotted line from the rotation center to the cursor previews the axis of rotation. Click to define the axis of rotation.
7. As you move the cursor, a preview of the rotated object displays. Click to set the rotation angle.

   The original object or its duplicate is rotated to the new position.



Rotating an object using the Standard and Standard Rotation modes

Rotating Objects

Rotation by Alignment

To rotate an object by aligning it with another object:

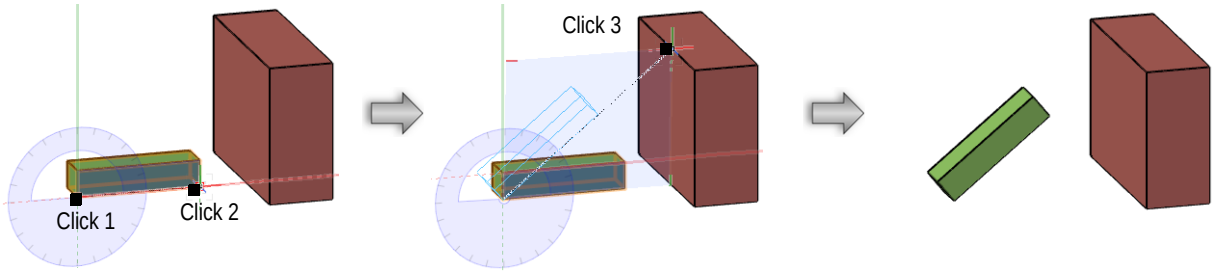
1. Select the object to rotate.
2. Select the **Rotate** tool from the Basic palette.

   To select additional objects or to change the currently selected objects to rotate, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd is being pressed.
3. Select **Standard** or **Duplicate** from the Tool bar, depending on whether the original or a duplicate object is to be rotated.
4. In a 3D view, select **Alignment Rotation** from the Tool bar.
5. A protractor feedback graphic displays around the cursor. Position the protractor on the appropriate rotation plane, and click at the center point of the rotation.

   The protractor graphic displays only when an object is selected.
6. The protractor graphic remains at the click point. As you move the cursor, the protractor rotates, and a dotted line from the rotation center to the cursor previews the axis of rotation. Click to define the axis of rotation.
7. As you move the cursor, a preview of the rotated object displays. Click the desired point to align the object.

   The original object or its duplicate is rotated to the new position.

   For the alignment and preview to work properly, all three clicks must be positioned on a snap point.



Rotating an object using the Standard and Alignment Rotation modes

Rotating Objects
Setting Snapping Parameters
Drawing with Snapping

Preset Rotation Angles

The rotate commands provide a number of ways to rotate objects in the screen plane without changing their base location in the drawing. Select one of the preset rotation angles, or specify a custom rotation angle.

To rotate at a specified angle:

1. Select the object(s) to rotate.
2. Select **Modify > Rotate**, and then the pre-set rotation.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the object(s), and select **Rotate** from the object context menu.

Click to show/hide the parameters.

Parameter	Description
Rotate Left 90º	Rotates the selected object counter-clockwise 90º
Rotate Right 90º	Rotates the selected object clockwise 90º
Flip Horizontal	Flips the selected object about its center
Flip Vertical	Flips the selected object vertically about its center

Rotating Objects

Custom Rotation

The center of rotation for this command is limited to the center of the object's bounding box. The rotation is performed within the screen plane.

To rotate by a custom angle:

1. Select the object or objects to rotate.
2. Select **Modify > Rotate > Rotate**. Alternatively, double-click the **Rotate** tool.

   The Rotate Object dialog box opens.
3. Enter the rotation angle.

   The precision of the values that can be entered (degrees, minutes, and/or seconds) depends on the Units setting.
4. Click **OK**.

Rotating Objects

3D Custom Rotation

The precision of the rotation values that can be entered (degrees, minutes, and/or seconds) depends on the Units setting.

To rotate at custom angles in 3D:

1. Select the 3D object or objects to rotate.
2. Select **Modify > Rotate > Rotate 3D**.

   The Rotate Object in 3D dialog box opens. Specify the rotation parameters.

Click to show/hide the parameters.

Parameter	Description
Rotation Angle	Enter the angle of rotation
Center of Rotation	Specify the desired center of rotation
Working Plane Center	Rotates using the center of the working plane
Object Center	Rotates about the center of the selected object
Next Mouse Click	Rotates around the next mouse click in the drawing window
Rotation Axis	Select whether the rotation axis will be based on the **Active Layer Plane** or the **Working Plane**, and about which axis

3. Click **OK**.

Rotating Objects

Unrotating 3D Objects

The **Unrotate 3D Objects** command returns any 3D object that has its own coordinate system to its upright position. Use the command to rotate symbols, extrudes, multiple extrudes, and sweeps so that they align with the world coordinate system.

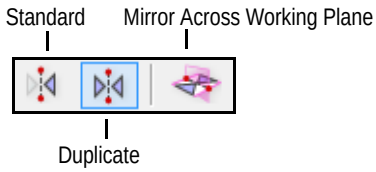
To unrotate a 3D object:

1. Select the 3D object or objects to unrotate.
2. Select **Modify > Unrotate 3D Objects**.

Rotating Objects

Mirroring Objects

The **Mirror** tool can mirror, or mirror and duplicate, objects across an axis line or across the working plane.



Mode	Description
Standard	Mirrors the selected object. The mirrored object is on the same plane as the original object.
Duplicate	Creates a duplicate of the selected object and then mirrors the duplicate. The mirrored object is on the same plane as the original object.
Mirror Across Working Plane (3D views only)	Mirrors the selected object to the other side of the working plane; available in 3D views

Click here for a video tip on this topic (Internet connection required).

Mirroring Objects Across an Axis
Mirroring Objects Across the Working Plane

Mirroring Objects Across an Axis

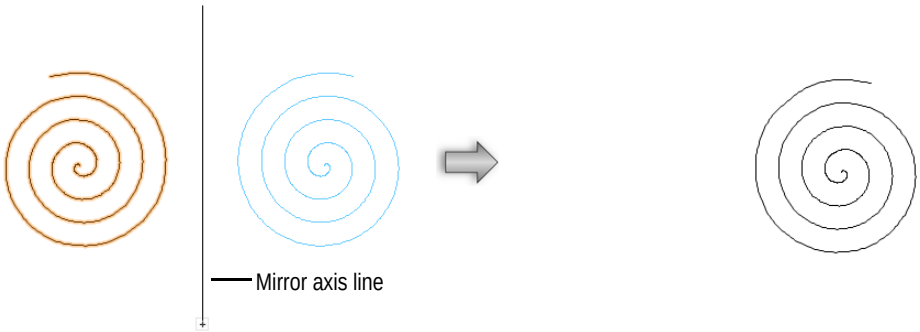
To mirror objects across an axis:

1. Select the object(s) to mirror.
2. Click the **Mirror** tool from the Basic palette.

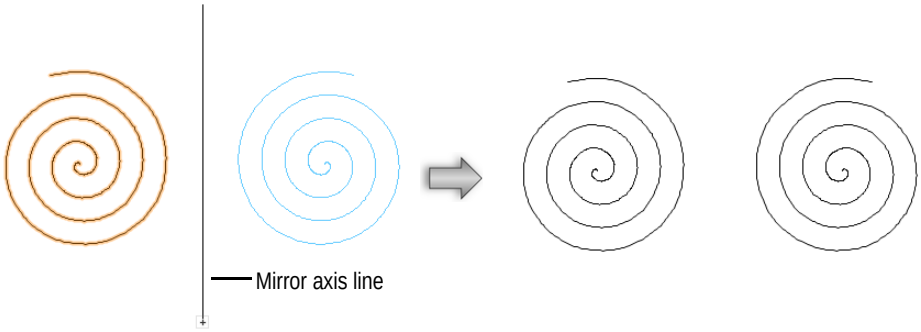
   To select additional objects or to change the currently selected objects to mirror, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd key is being pressed.
3. Select **Standard** or **Duplicate** from the Tool bar, depending on whether the original object is to remain in place.

4. Click to set the beginning of the axis line. As you begin to drag the mouse to create the axis line, a preview of the mirrored object's placement displays. Click again to set the end of the line. For objects in walls (such as doors or columns), the axis line is constrained to be perpendicular to the wall. 2D objects are previewed as vectors without fill and 3D objects are previewed in Wireframe mode, regardless of the current render mode.

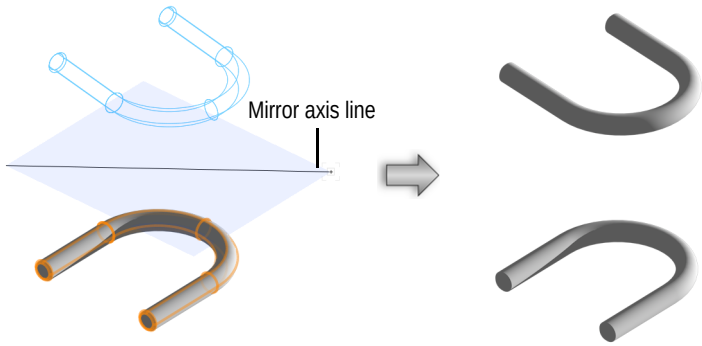
   The object or its duplicate is mirrored on the opposite side of the plane created by projecting the axis line onto the working plane.



Standard mode previews the item's mirrored placement on the other side of the axis line after the first click, then moves it at the second click

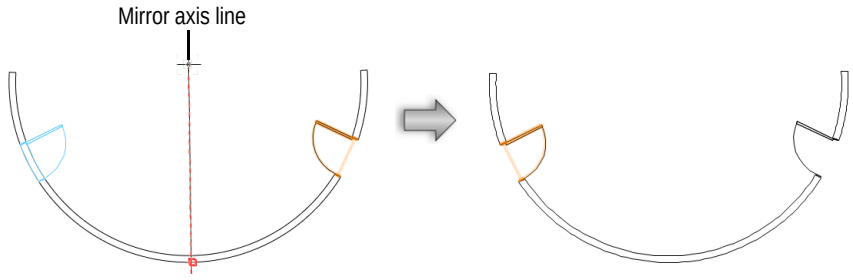


Duplicate mode previews the mirrored duplicate item's placement on the other side of the axis line after the first click, then places the new object at the second click



   The end points of the mirror axis line must occupy distinct points on the active working plane. In 3D views, mirroring will not work if the two points are coincident on the active plane.

For objects in walls, the mirrored object is placed in the wall, if there is room for it.



Duplicate mode creates a mirror duplicate of the door inserted into the wall

Mirroring Objects Across the Working Plane
Mirroring Objects

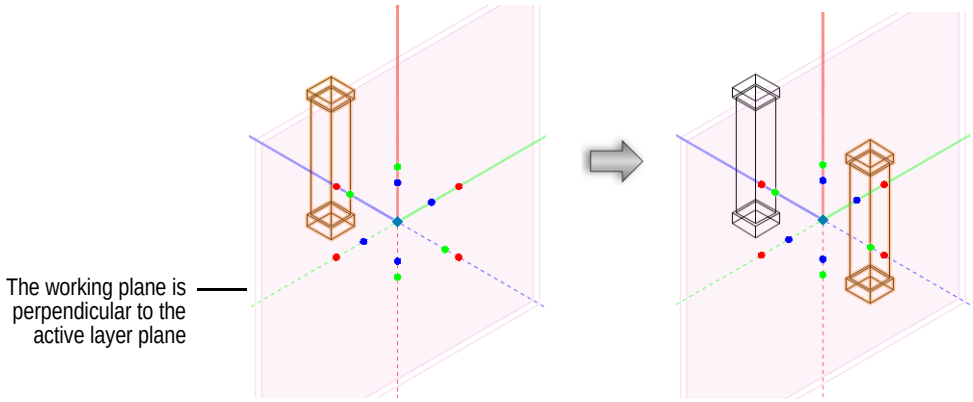
Mirroring Objects Across the Working Plane

To mirror objects across the working plane:

1. Select the object(s) to mirror.
2. Hybrid objects such as walls or columns must remain perpendicular to the active layer plane. If you are mirroring a hybrid object, set the working plane so that it intersects the active layer plane at 90 degrees.
3. Click the **Mirror** tool from the Basic palette.

   To select additional objects or to change the currently selected objects to mirror, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd key is being pressed.
4. Select **Standard** or **Duplicate** from the Tool bar, depending on whether the original object is to remain in place.
5. Click **Mirror Across Working Plane** mode.

   The object or its duplicate is mirrored to the opposite side of the working plane. There is no need to create an axis line.



Duplicate mode creates a mirror duplicate of the column across the working plane

Mirroring Objects Across an Axis
Mirroring Objects

Converting Objects

Objects can be converted in several ways.

Convert to Lines

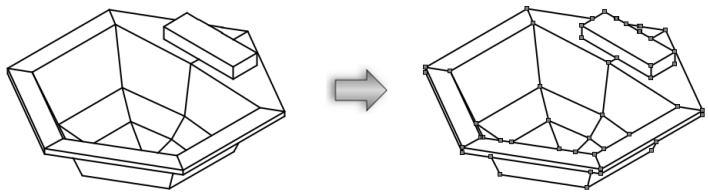
The **Convert to Lines** command changes a single object into the series of lines needed to create it. For example, a rectangle is converted into four lines.

Circles and ovals can also be converted into numerous line segments. However, especially when converting circles, the accuracy of the line segments depends on the **2D conversion resolution** setting chosen in the Vectorworks Preferences dialog box.

To convert an object to lines:

1. Select the object to convert.
2. Select **Modify > Convert > Convert to Lines**.
3. If the object can be rendered, select the wireframe, hidden line, or dashed hidden line rendering conversion option.
4. Click **OK**.

   The object is converted into lines and each segment can be manipulated.



Convert Copy to Lines

The **Convert Copy to Lines** command works like the **Convert to Lines** command except that it first makes a copy of the object and then converts the copy into line segments. The original object remains intact.

To convert a copy of an object to lines:

1. Select the object to convert.
2. Select **Modify > Convert > Convert Copy to Lines**.
3. If the object can be rendered, select the wireframe, hidden line, or dashed hidden line rendering conversion option.
4. Click **OK**.

   A copy of the object is converted into lines and each segment can be manipulated.

Convert to Lines

Convert to Polygons

The **Convert to Polygons** command changes any object with a surface into the 2D polygon or series of polygons needed to create it.

Circles and ovals can also be converted. However, especially when converting circles, the accuracy of the polygons depends on the **2D conversion resolution** setting chosen in the Vectorworks Preferences dialog box.

When most closed 3D geometry is converted to 2D polygons with the hidden line rendering option, back-facing polygons are discarded.

To convert an object to polygons:

1. Select the object to convert.
2. Select **Modify > Convert > Convert to Polygons**.
3. If the object can be rendered, select either the wireframe or hidden line rendering conversion option.
4. Click **OK**.

   The object is converted to a group of polygons. To select an individual polygon, first select **Modify > Ungroup**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Convert to 3D Polygons

Convert Copy to Polygons

The **Convert Copy to Polygons** command works like the **Convert to Polygons** command, except that it first makes a copy of the object and then converts the copy into 2D polygons. The original object remains intact.

To convert a copy of an object to polygons:

1. Select the object to convert.
2. Select **Modify > Convert > Convert Copy to Polygons**.
3. If the object can be rendered, select either the wireframe or hidden line rendering conversion option.
4. Click **OK**.

   A copy of the object is converted to a group of polygons. To select an individual polygon, first select **Modify > Ungroup**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Convert to Polygons

Convert to 3D Polygons

The **Convert to 3D Polys** command converts any 2D line or surface object, including polylines, polygons, circles, ovals, and rectangles, into a 3D polygon. Once converted, the new polygon contains a Z dimension, assigning it a place in 3D space. It can be rotated and manipulated with 3D tools. The new polygon will not, however, have a thickness.

When you convert two or more objects simultaneously, the converted objects are placed in a group.

To convert a 2D object to 3D polygons:

1. Select the 2D object or objects to convert.
2. Select **Modify > Convert > Convert to 3D Polys**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Convert to Polygons
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Convert to Mesh

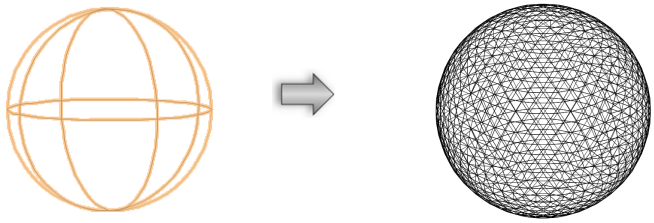
The **Convert to Mesh** command converts any extrude, multiple extrude, sweep, or wireframe object into a mesh object. Use the Object Info palette to edit the mesh object vertices. The command can also collect a number of separate 3D polygons into a single mesh object.

A mesh object can contain up to 30,000 vertices.

To convert to mesh:

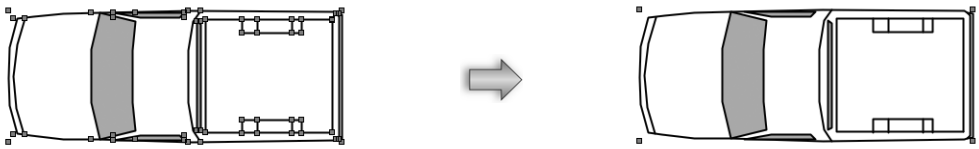
1. Select the 3D object to convert.
2. Select **Modify > Convert > Convert to Mesh**.

   The object changes to a mesh object. To edit the mesh, select **Modify > Edit Group** and edit the 3D polygons that make up the mesh.



Grouping Objects

The **Group** command groups two or more individual objects (including text and symbols) together. The group of objects is then treated as a single object. For example, grouped objects can be moved to a different layer in one move. In addition, this command can group two or more groups of objects into a single group.



Individual objects

Grouped objects

To group objects:

1. Select the objects (or groups) to be grouped.
2. Select **Modify > Group**.

   The objects are grouped into one object, and the group is placed in the active class.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing a Group
Ungrouping
Returning to Top Level

Editing a Group

Use the **Modify > Edit Group** command to edit individual objects in a group without ungrouping them. This command also edits a group of objects that is nested inside another group; each time the command is selected, the program moves one level deeper into the group. See “Object Editing Mode” on page 1004 for details.

Returning to Top Level

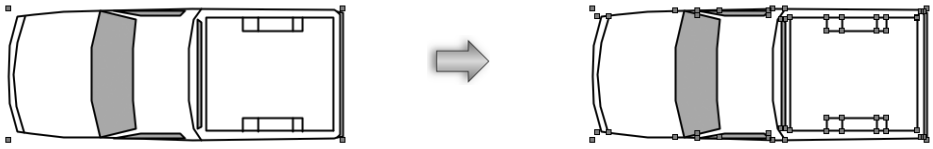
Use the **Top Level** command when editing nested groups, to return to the main drawing area in a single step.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Select **Modify > Top Level**.

The program automatically exits all groups being edited and returns to the main drawing area.

Ungrouping

The **Ungroup** command ungroups objects or groups that were previously combined with the **Group** command. Nested groups must be ungrouped one at a time.



Grouped objects

Ungrouped objects

To ungroup a group of objects:

1. Select the group to ungroup.
2. Select **Modify > Ungroup**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the group, and select **Ungroup** from the object context menu.
3. A plug-in object will lose its plug-in functionality if it is ungrouped. Confirm that the high-level objects should be ungrouped.

   Also, if the group has attached records, you are prompted to attach the records to the ungrouped objects. Click **Yes** to transfer the records to each ungrouped item; click **No** to discard the record information.

   If you transfer records to multiple ungrouped objects, you will have duplicates. This may be a problem if the original grouped object was being tracked on a worksheet, for example.

Aligning and Distributing Objects

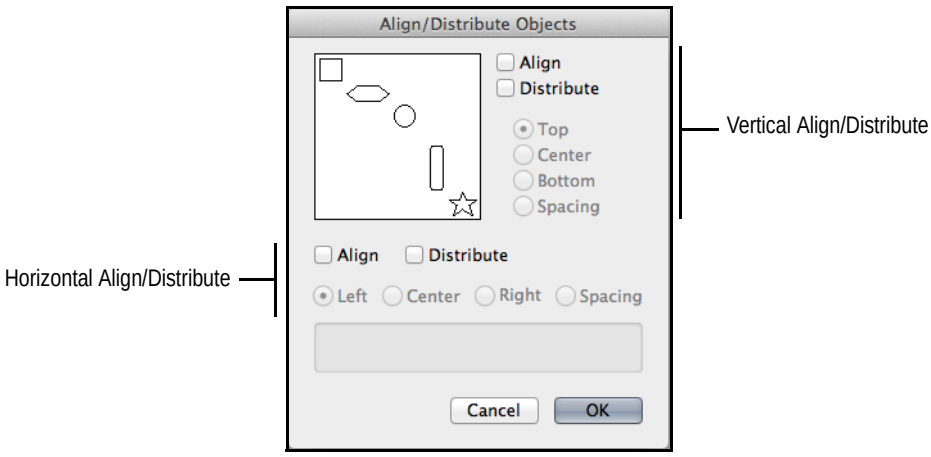
Aligning and Distributing Objects in the Screen Plane

The **Align/Distribute** command aligns and distributes multiple objects in the screen plane. Alignment and distribution is based upon the vertical and horizontal axes, relative to the screen.

To align/distribute objects:

1. Select the objects to be aligned/distributed.
2. Select **Modify > Align > Align/Distribute**. The Align/Distribute Objects dialog box opens.

   Alternatively, Right-click (Windows) or Ctrl-click (Mac) on one of the selected objects and select **Align/Distribute** from the context menu. The most common align and distribute commands display on a flyout context menu. Select the desired command, or select **Align/Distribute** to open the Align/Distribute Objects dialog box.



3. Select the object alignment/distribution criteria. Objects are only moved along the horizontal and vertical axes.

Click to show/hide the parameters.

Parameter	Description
Align	Select to align items along the horizontal or vertical axis
Distribute	Select to distribute items along the horizontal or vertical axis
Horizontal	
Left	Align/distribute by the left side of the selected objects
Center	Align/distribute by the centers of the selected objects
Right	Align/distribute by the right side of the selected objects
Spacing	Distribute the spacing equally between the left and right sides of adjacent objects
Vertical	
Top	Align/distribute by the top of the selected objects
Center	Align/distribute by the centers of the selected objects
Bottom	Align/distribute by the bottom of the selected objects
Spacing	Distribute the spacing equally between tops and bottoms of adjacent objects

4. Click **OK**.

   The program aligns/distributes the selected objects.

Loci and locked objects are special objects and behave differently when present during an alignment/distribution operation.

If a locus point is one of the selected objects, all objects are aligned relative to that locus. If there are multiple loci in the selection, then the loci are aligned/distributed like any other object.

Locked objects in a selection do not move. Other objects are aligned/distributed relative to the locked objects.

Aligning and Distributing Objects in 3D
Aligning Objects to Grid
Aligning and Distributing Leader Lines

Aligning and Distributing Objects in 3D

Objects can be aligned and/or distributed in relation to each other in 3D space. 3D object alignment/distribution is based on either the ground or working plane.

The **Align/Distribute 3D** command aligns and distributes multiple objects. Alignment and distribution is based upon the X, Y, and Z axes of the active layer plane or the X', Y', and Z' axes of the working plane.

Keep in mind the following points about this command:

- 3D loci and locked objects are special objects and behave differently when present during an alignment/distribution operation.
- If a 3D locus point is one of the selected objects, all objects are aligned relative to that locus. If there are multiple loci in the selection, then the loci are aligned/distributed like any other object.
- Locked objects in a selection do not move. Other objects are aligned/distributed relative to the locked objects.
- 2D screen objects are ignored by the **Align/Distribute 3D** command.

To align/distribute objects in 3D:

1. Select the objects to be aligned/distributed.
2. Select **Modify > Align > Align/Distribute 3D**.

   The 3D Object Align and Distribute dialog box opens. Select the alignment/distribution criteria.

Click to show/hide the parameters.

Parameter	Description
Alignment/ Distribution Axes	Select whether to align/distribute about the active layer plane or the working plane
Align/Distribute	Select **Align** and specify the coordinate value to use when aligning objects, or select **Distribute** to distribute objects
Minimum	Align/distribute objects along the specified axis by the minimum coordinate value of each object
Center	Align/distribute objects along the specified axis by the center coordinate value of each object
Maximum	Align/distribute objects along the specified axis by the maximum coordinate value of each object
Spacing	Distribute spacing between objects along the specified axis, ensuring that the space between the objects is equal along the axis

3. Click **OK**.

   The program aligns/distributes the selected objects.

Aligning and Distributing Objects in the Screen Plane
Aligning Objects to Grid

Aligning Objects to Grid

The **Align to Grid** command aligns objects to the snap grid, placing the upper-left corner of each object at its closest grid point. The **Align to Grid** command can be used to realign objects to the snap grid when the grid's settings have changed. It also aligns objects to the grid if they were either created with **Snap to Grid** snapping turned off or were moved off the grid after they were created.

To align an object to a grid:

1. Select the object or objects to align.
2. Select **Modify > Align > Align to Grid**.

   The program aligns the object(s), placing the upper-left corner of each object at its closest grid point.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Aligning and Distributing Objects
Aligning and Distributing Leader Lines
Snap and Reference Grids

Aligning and Distributing Leader Lines

Many Vectorworks objects include tags with leader lines that point to specific items in a drawing. For drawings that have several such objects, you can align the leader lines to improve readability. The **Align/Distribute Leader Lines** command works on the following objects:

Vectorworks Fundamentals objects
- Callouts

Vectorworks Design Series objects
- Detail bubbles
- Elevation benchmarks
- Existing trees
- Hardscapes
- Landscape areas
- Leader lines
- Plants
- Redlines
- Stakes

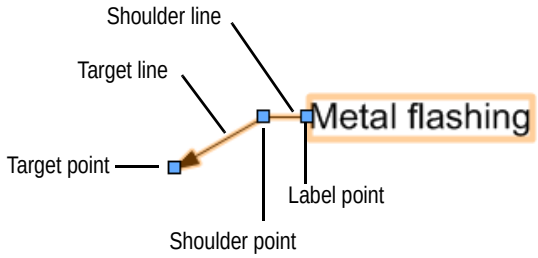
To align leader lines:

1. Select the objects whose leader lines you want to align and/or distribute.
2. Select **Modify > Align > Align/Distribute Leader Lines**.
   - If none of the selected objects can be processed by the command, a message displays, and the command is canceled.
   - If one or more of the selected objects cannot be processed, a different message displays; you can either continue to align the remaining objects, or cancel the command from the dialog box that displays next.
3. The Align Leader Lines dialog box displays.

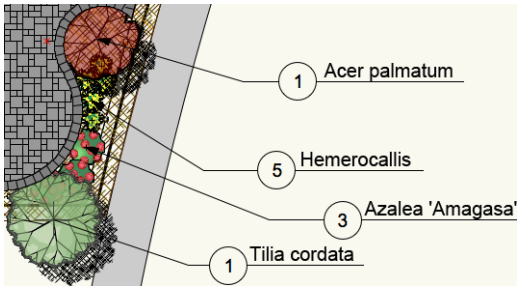
Click to show/hide the parameters.

Parameter	Description
Horizontal Alignment	
Align shoulder points	Select to align the shoulder points of the selected leader lines
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

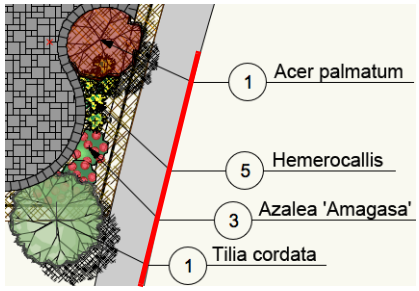
Parameter	Description
Align label points	Select to align the label points of the selected leader lines. If this option is used in conjunction with **Align shoulder points**, all of the selected label points are also aligned to the label point of the topmost leader line.
Move shoulder points	If **Align shoulder points** is not selected, and **Align label points** is selected, select this option to move both the label and shoulder points, keeping the current distance between those points for each line. Deselect this option to move only the label points, keeping the current shoulder point positions. This setting does not apply to elevation benchmarks without an offset.
Vertical Distribution	
None	Select to leave the selected leader lines in the same vertical position when they are aligned horizontally
Parallel objective lines	Select to move the shoulder points vertically, so that the target lines of all selected objects are parallel to the line of the topmost object. This option can be used with or without a horizontal alignment option. This setting does not apply to elevation benchmarks.
Equal distance between shoulder lines	Select to move the shoulder points vertically, so that the vertical distance between the shoulder lines of all selected objects is equal to the distance between the lines of the two topmost objects. This option can be used with or without a horizontal alignment option. Offsets may be added to elevation benchmarks or adjusted as needed.



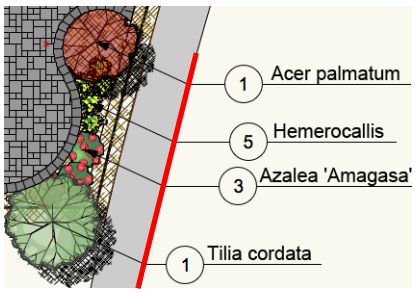
4. Select the desired alignment options and click **OK**.
5. If one of the horizontal alignment options was selected, click and drag to draw a reference line along which to align the selected shoulder or label points; the line must not be horizontal. Click again to complete the reference line.
6. The objects are aligned as specified.



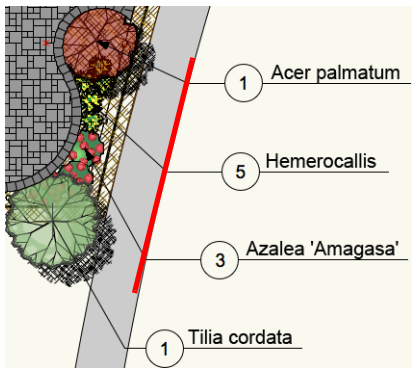
Unaligned plant labels



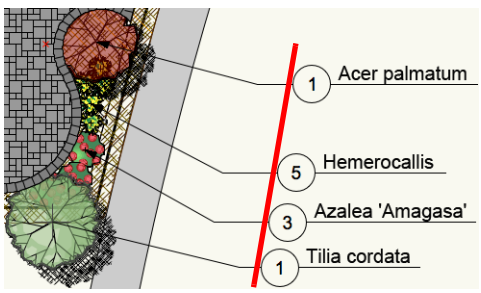
Labels aligned with **Align shoulder points** (along red reference line), with no vertical distribution



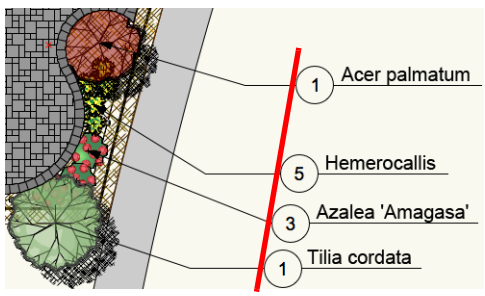
Labels aligned with **Align shoulder points** (along red reference line), with **Parallel target lines**



Labels aligned with **Align shoulder points** (along red reference line), with **Equal distance between shoulder lines**



Labels aligned with **Align label points** (along red reference line), and **Move shoulder points**



Labels aligned with **Align label points** (along red reference line), but no **Move shoulder points**

Aligning and Distributing Objects in the Screen Plane
Aligning and Distributing Objects in 3D
Aligning Objects to Grid

Extruding Objects

Single Extrude

The **Extrude** command changes 2D objects into 3D objects with a height. The type of object and its attributes determines the type of extrude that is produced. Lines are extruded as flat planes, while all other objects are extruded as wireframe 3D objects. While more than one object can be extruded at a time, grouped objects must first be ungrouped in order to be extruded. Objects that are locked must first be unlocked in order to be extruded.

Objects are extruded into the current viewing projection, so the extrusion direction is dependent on the current view. They can be rotated afterwards if desired. See “Rotate Tool” on page 1022 or “Preset Rotation Angles” on page 1024. The extruded object height can be edited interactively with the **Reshape** tool; see “Reshaping Extruded Objects and Solid Primitives” on page 1052 for more information.

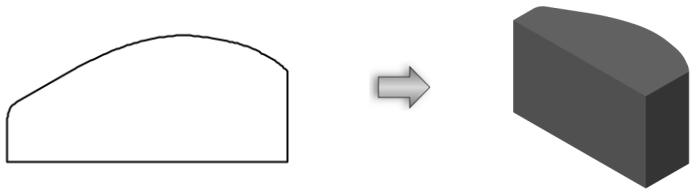
When you extrude multiple objects at the same time, they are turned into a single 3D group of objects. To edit an individual object, either ungroup the set or use the **Edit Extrude** command to access the individual objects.

The **Push/Pull** tool can also be used to create extruded objects from planar objects; see “Direct Modeling with the Push/Pull Tool” on page 357.

To extrude an object:

1. Select the 2D object(s) to extrude.
2. Select **Model > Extrude**.

   The Create Extrude dialog box opens.
3. Enter the **Extrusion** height and specify any change in size along the X and Y axes, and then click **OK**.



Multiple Extrude
Extrude Along Path
Creating a Tapered Extrude
Face-based Addition or Subtraction

Multiple Extrude

A pyramid, sphere, or other 3D object can be created from a series of 2D objects using the **Multiple Extrude** command. Include a 2D locus, or several loci, in the selection to provide an extrusion reference point or points.

The process matches segments on each object to create the extrude. The segment order may be reversed for one or more of the objects, producing unexpected results (such as a twisted extrusion). If this occurs, convert the objects to open polygons prior to selecting the **Multiple Extrude** command.

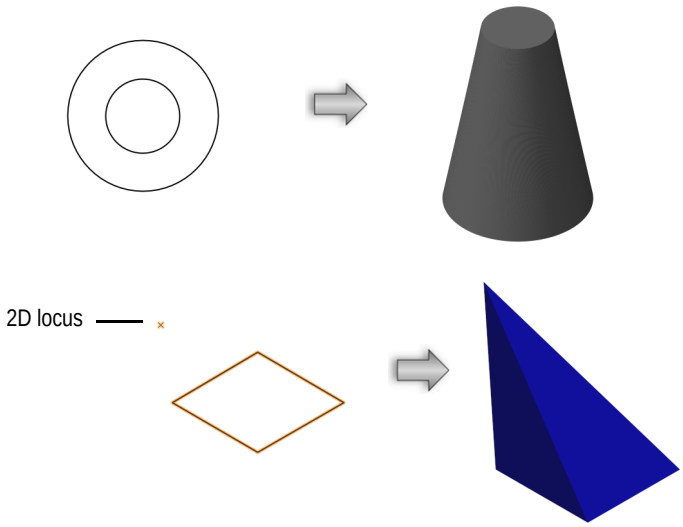
To create a multiple extrude:

1. Select the 2D objects to extrude.
2. Select **Model > Multiple Extrude**.

   The Create Extrude dialog box opens.
3. Enter the **Extrusion** height and specify the change in size along the X and Y axes.

4. Click **OK**.

   The program extrudes the set of objects/locus points by connecting one to another in their stacking order, turning the objects into a single 3D object.



Single Extrude
Extrude Along Path
Creating a Tapered Extrude

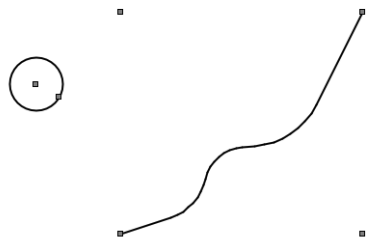
Extrude Along Path

The **Extrude Along Path** command extrudes profile objects along a given path. Profile objects can be 2D objects, 3D polygons, and NURBS curves. Profile objects cannot be non-planar, self-intersecting, or a mixture of 2D and 3D profiles. If the path is not a NURBS curve, it is converted into a NURBS curve during this operation.

When you use the **Edit Extrude Along Path** command on a 3D object created using the **Extrude Along Path** command, a dialog box opens with the choice to either edit the original path or profile object.

To extrude along a path:

1. Select the object to be extruded, and the object to use as the path.



2. Select **Model > Extrude Along Path**.

   The Extrude Along Path dialog box opens. Specify the extrusion parameters.

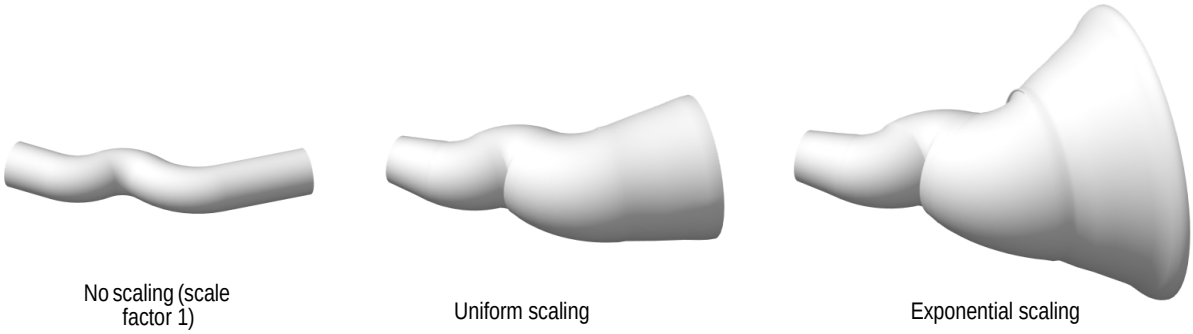
Click to show/hide the parameters.

Parameter	Description
Select a path object	Click the **Next** or **Prev** button to select the object that represents the path

Parameter	Description
Uniform Scale	Scales the object linearly along the designated path; enter a **Scale Factor** of 1 to perform no scaling, or enter a positive scale factor other than 1 for uniform scaling
Exponential Scale	Scales the object exponentially along the designated path; enter a **Shape Factor** of 0 to perform no scaling, or enter a shape factor other than 0 for exponential scaling
Lock Profile Plane	Locks the orientation of the profile plane to the global Z axis, constraining the extrusion to be perpendicular to the XY plane
Fix Profile	Maintains the original relative position and orientation of the profile object with respect to the path. Normally, this should be deselected, so that the profile is moved and rotated with respect to the path.

Scaling options are available when the path object is a single continuous curve without sharp corners or discontinuities.

3. Click **OK** to extrude the object along the selected path.



4. The type of **Scale**, **Shape/Scale Factor**, **Lock Profile Plane**, and **Fix Profile** status of an extrude along path object can be edited in the Object Info palette.

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Single Extrude
Multiple Extrude
Creating a Tapered Extrude

Creating a Tapered Extrude

A tapered extrude can be created from 2D objects, 3D polygons, and NURBS curves, circles, and arcs. (2D objects cannot be screen plane objects; change them to planar objects first. See “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152.)

This command easily creates wavy extrusions by entering a taper value of zero.The command also extrudes 2D objects into 3D objects with a defined taper. Use it to convert a single object or several objects at the same time. Once created, the height and taper can be edited in the Object Info palette.

The tapered extrude object height can be edited interactively with the **Reshape** tool; see “Reshaping Extruded Objects and Solid Primitives” on page 1052 for more information.

To create a tapered extrude:

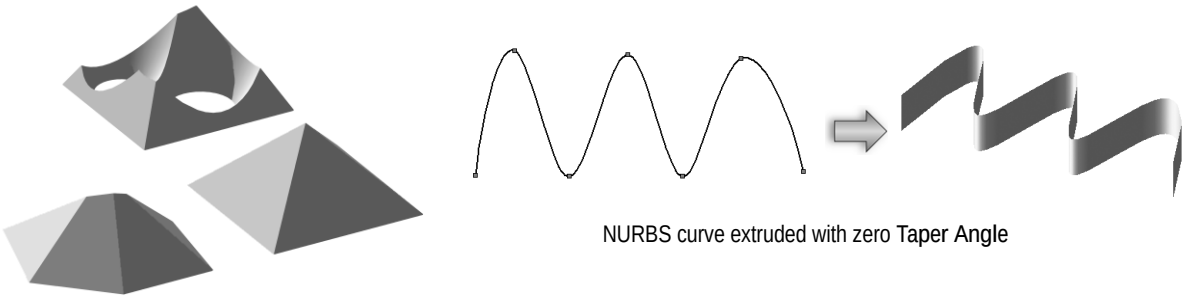
1. Select the object(s) to turn into a tapered extrude.
2. Select **Model > Tapered Extrude**.

   The Create Tapered Extrude dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Enter the **Height (Z)** of the extrude and the **Taper Angle**. A positive taper degree shrinks the object as it extrudes, while a negative taper degree widens the object.

   Set a **Taper Angle** of 0 to create an extrusion with no taper.

4. Click **OK** to create the tapered extrude.



If a tapered extrude fails, the object reverts to the last known "good" value.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Single Extrude
Multiple Extrude
Extrude Along Path
Face-based Addition or Subtraction

Sweeping Objects

The **Sweep** command converts 2D objects into 3D cylindrical objects. Use it to convert a single object at a time, or to convert several selected objects. While more than one object can be swept at a time, grouped objects must first be ungrouped. Locked objects must first be unlocked in order to create a sweep.

A sweep has four basic elements: a centroid (locus), an arc angle, a segment angle, and a pitch. The locus acts as the sweep's center of rotation. If a locus is not selected, the program sweeps the object around its left edge or point farthest on the left if more than one object is selected. A locus can be relocated or added after creation using the **Edit Sweep** command. The other three elements (arc angle, segment angle, pitch) can be edited after the sweep has been created through the Object Info palette.

If more than one object is selected when creating a sweep, the objects are automatically grouped. To edit an object within the sweep, use the **Edit Sweep** command.

To create a sweep:

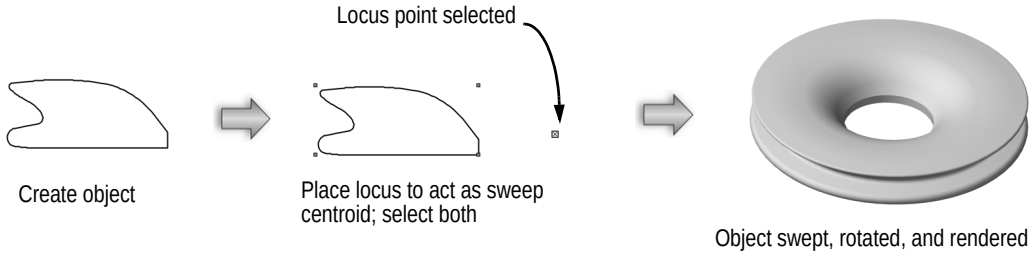
1. Select the 2D object(s) to sweep and, if created, the locus.
2. Select **Model > Sweep**.

   The Create Sweep dialog box opens. Enter the sweep criteria and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Size	
Height	Indicates the sweep height
Radius	Indicates the sweep radius
Angles	
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Start Angle	Specifies the angle to begin the sweep; normally start at 0
Arc Angle	Specifies the degree of the sweep. By default, objects are swept a full 360º.
Segment Angle	Indicates the number of segments that make up the sweep. The default is 36 segments, each one 10º from the next on the arc. The angle setting must be a positive number. A large number of segments can slow down performance.
Pitch	Sets the degree to which the sweep spirals. A sweep's pitch is represented in the plus- or minus-height per revolution. For example, if the pitch is 1", every revolution raises the object a single inch. The default is 0.



Reshaping Objects

The **Reshape** tool reshapes an object after it has been created, by moving, removing, changing, or adding to its vertices. Change the length of objects (including dimensions), reshape single objects, or reshape multiple objects at once. Reshape polygons and polylines, including lines drawn with the **Freehand** tool (which are considered polylines). In addition, specify exact radius measurements for circular arc control points. The **Reshape** tool can also be used to reshape 3D solids, walls, roofs, NURBS curves and surfaces, and other objects.

The modes and functionality available for the **Reshape** tool depend on the currently selected object or objects, and the current view.

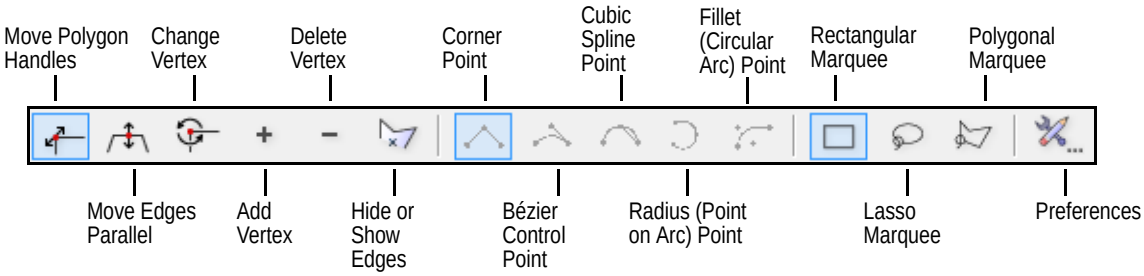
Functionality Enabled	Selection or View
2D reshape	2D planar or screen object(s), in any view
2D reshape	Multiple objects selected, in any view, or a reshape marquee exists
3D reshape	Single 3D polygon, 3D solid, NURBS curve, NURBS surface, section line (Vectorworks Design Series required), in any view
3D reshape	Single wall, roof object, plug-in object with custom reshape behavior (such as the retaining wall site modifier, Vectorworks Design Series required), in a 3D view
2D reshape	Wall, roof object, plug-in object in Top/Plan view
Marquee	No selection

Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2D Reshape Modes

The **Reshape** tool has six reshape modes when 2D functionality is enabled. It also has five control point modes for changing the existing vertex. In addition, the tool can operate in three marquee selection modes.



Mode	Description
Move Polygon Handles	Changes a shape by clicking and moving a vertex or midpoint handle
Move Edges Parallel	Moves an edge parallel to its original position without changing the adjacent angles
Change Vertex	Changes an existing vertex into a corner, Bézier, cubic, radius, or arc vertex
Add Vertex	Adds a vertex as a corner, Bézier, cubic, radius, or arc vertex
Delete Vertex	Removes the selected vertex
Hide or Show Edges	Clicking on the midpoint hides the polyline or polygon segment; clicking on a vertex hides the segment that follows the vertex
Corner Point	In Change Vertex mode, changes the vertex to a corner vertex; in Add Vertex mode, adds a corner vertex
Bézier Control Point	In Change Vertex mode, changes the vertex to a Bézier vertex; in Add Vertex mode, adds a Bézier vertex
Cubic Spline Point	In Change Vertex mode, changes the vertex to a cubic vertex; in Add Vertex mode, adds a cubic vertex
Radius (Point on Arc) Point	In Change Vertex mode, changes the vertex to a radius vertex; in Add Vertex mode, adds a radius vertex
Fillet (Circular Arc) Point	In Change Vertex mode, changes the vertex to an arc vertex; in Add Vertex mode, adds an arc vertex. Click **Preferences** from the Tool bar, enter a **Fillet Radius** in the Fillet Settings dialog box, and click **OK** (for the largest possible fillet, set the **Fillet Radius** to zero).
Rectangular Marquee	Creates a rectangular marquee box around several vertices. Click to set the start point, drag the mouse in the desired direction, and release to set the end point. All vertices within the marquee are selected for performing a subsequent reshape operation.
Lasso Marquee	Creates a free-form marquee around several vertices, allowing a more exact selection of vertices of irregular 2D shapes. Click to set the start point, drag the mouse in the desired direction, and release to set the end point. All vertices within the marquee are selected for performing a subsequent reshape operation.
Polygonal Marquee	Creates a marquee with an irregular polygonal shape around several vertices. Click to set the start point, and then continue clicking to define the shape. Double-click to finish the marquee. All vertices within the marquee are selected for performing a subsequent reshape operation.

In addition to clicking the **Reshape** tool from the Basic palette, there are other ways to reshape 2D objects.

- Double-clicking eligible 2D objects (polylines, polygons, rectangles, rounded rectangles, and ovals) with the **Selection** tool automatically activates the **Reshape** tool.
- 2D objects and symbols can be resized with the Single Object Interactive Scaling mode of the **Selection** tool; see "The Selection Tool" on page 109.
- Certain objects (polylines, polygons, rectangles, rounded rectangles, circles, ovals, and arcs) can also be reshaped with the **Freehand** tool. See "Reshaping Objects with the Freehand Tool" on page 301.

  Additional editing modes are available for certain other objects, such as a site model (Vectorworks Design Series required). Those specific modes are described in sections about editing or reshaping the object.

Functionality similar to the **Reshape** tool is available from the Object Info palette Shape tab; see "Editing Vertex-Based Objects" on page 1002.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

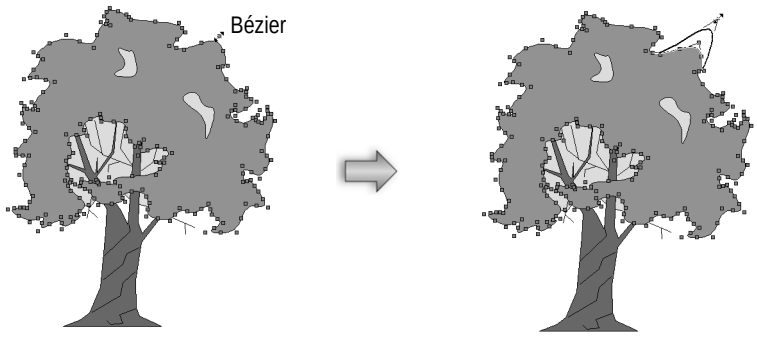
Move Polygon Handles Mode
Move Edges Parallel Mode
Change Vertex Mode
Add Vertex Mode
Delete Vertex Mode
Hide or Show Edges Mode
Performing Multiple Reshapes
The Selection Tool
Smoothing Objects

Move Polygon Handles Mode

Select this mode to reshape a 2D rectangle, rounded rectangle, oval, polyline, polygon, or path object by moving one of its vertices or midpoint handles.

 To reshape a 2D object by moving a vertex or midpoint handle:

1. Select the 2D shape to change.
2. Click the **Reshape** tool from the Basic palette.
3. Click the **Move Polygon Handles** mode from the Tool bar.
4. Click the vertex or handle to move. When the Reshape cursor displays, move the mouse to the new location.
5. Click to place the vertex/handle.



If an object cannot be reshaped with this tool, its handles do not display.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Performing Multiple Reshapes
Reshaping Objects
Editing Vertex-Based Objects

Move Edges Parallel Mode

Select this mode to reshape a 2D rectangle, rounded rectangle, oval, polyline, polygon, or path object by moving edges without changing the adjacent angles.

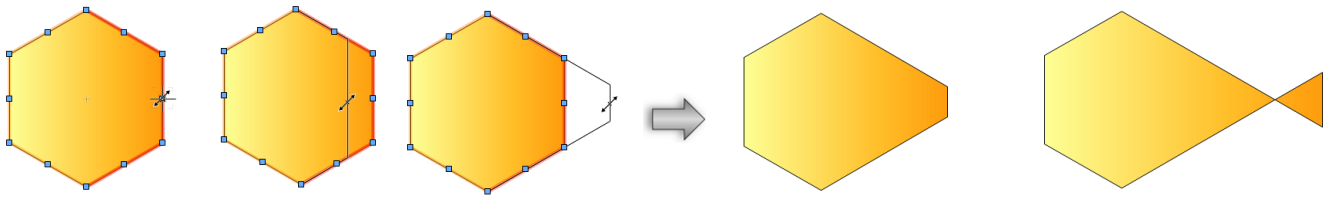
To reshape a 2D object by moving edges:

1. Select the 2D shape to change.
2. Click the **Reshape** tool from the Basic palette.
3. Click the **Move Edges Parallel** mode from the Tool bar.

   Reshape handles appear at the midpoint of each editable edge and on each editable corner vertex. Only straight edges with straight adjacent segments can be reshaped with this mode; if an object or segment of an object cannot be reshaped, its handles do not display.
4. Click the reshape handle to move. When the Reshape cursor displays, move the mouse to the new location.

   To offset the edges adjacent to the corner vertex by the same distance, hold the Option key (Mac) or Alt key (Windows) when moving the vertex.
5. Click to place the edge/vertex.

Moving a midpoint handle



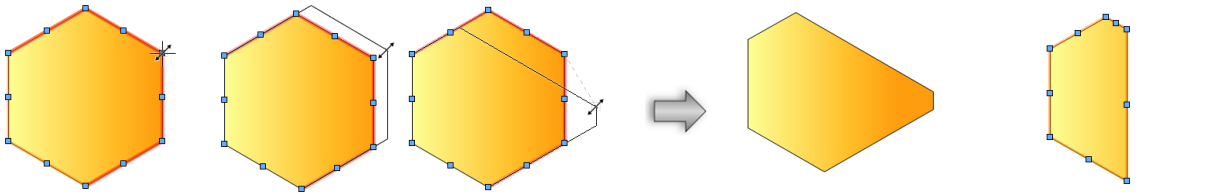
The edge to move and adjacent edges are highlighted

The new shape previews

The moving edge changes length to maintain the adjacent angles

If the edge is moved beyond the angles' intersection point, a self-intersecting polygon is created

Moving a corner vertex



The two adjacent edges on each side of the vertex are highlighted

The new shape previews

The edges adjacent to the vertex move parallel; the next edges extend to maintain the angle between the edges

If two or more vertices become coincident, the extra vertices are removed

Move Edges Parallel mode also reshapes open polygons; if the length of an open edge becomes zero, the extra vertices are removed and the shape becomes a closed polygon.

Reshaping Objects
Editing Vertex-Based Objects

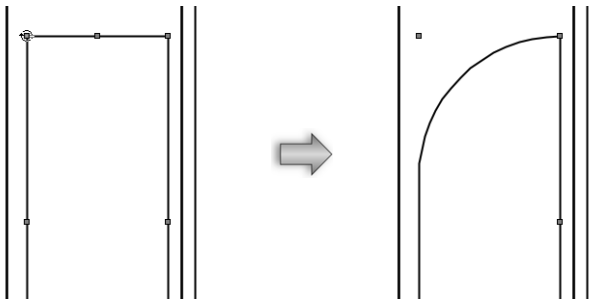
Change Vertex Mode

Select this mode to change an existing vertex to a different type of control point: corner, Bézier, cubic, radius, or fillet (circular arc) point.

To reshape a 2D polyline or polygon by changing a vertex:

1. Select the polyline or polygon to change.
2. Click the **Reshape** tool from the Basic palette.
3. Click the **Change Vertex** mode from the Tool bar.
4. From the Tool bar, select the new type of control point for the vertex.

   Change vertex does not work on a midpoint handle. Select only a vertex point.
5. Click on a vertex to change it to the new type.



Click on the vertex when the Change Vertex cursor displays

Reshaping Objects
Converting Polyline Vertices from Arc to Radius
Editing Vertex-Based Objects

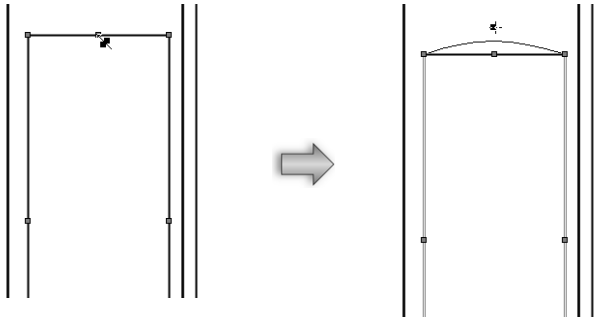
Add Vertex Mode

Select this mode to add a vertex to the polygon/polyline. The vertex can be any of the following: corner, Bézier, cubic, radius, or fillet (circular arc).

To reshape a 2D polyline or polygon by adding a vertex:

1. Select the polyline or polygon to change.
2. Click the **Reshape** tool from the Basic palette.
3. Click the **Add Vertex** mode from the Tool bar.
4. From the Tool bar, select the type of control point for the new vertex.
5. Move the cursor to an existing vertex near the location where the new vertex is to be added. The cursor becomes an arrow with two black diamonds when a vertex can be added.

   Clicking on the handle at a center point along an edge adds a new vertex exactly at that center point.
6. Drag the cursor to the desired location for the new vertex.
7. Click at the new location.



With the Add Vertex cursor, click-drag to the new location

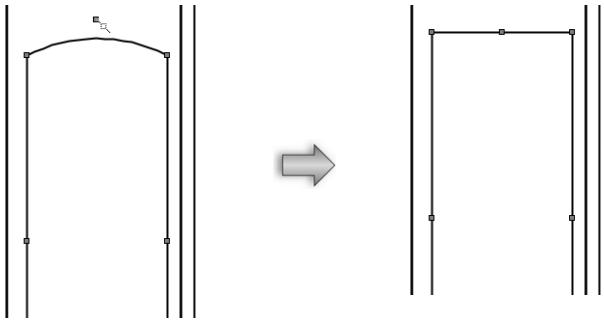
Reshaping Objects
Editing Vertex-Based Objects

Delete Vertex Mode

An existing vertex on the polygon/polyline can be deleted.

To reshape a 2D polyline or polygon by deleting a vertex:

1. Select the polyline or polygon to change.
2. Click the **Reshape** tool from the Basic palette.
3. Click the **Delete Vertex** mode from the Tool bar.
4. Click on an existing vertex to delete it.



With the Delete Vertex cursor, click the vertex to remove it

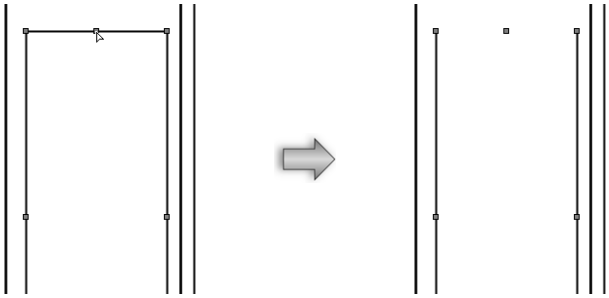
Reshaping Objects
Editing Vertex-Based Objects

Hide or Show Edges Mode

Select this mode to hide, but not delete, a portion of the object's boundary.

To hide or show the edge of a 2D polyline or polygon:

1. Select the polyline or polygon to change.
2. Click the **Reshape** tool from the Basic palette.
3. Click the **Hide or Show Edges** mode from the Tool bar.
4. Click a vertex near the edge to be hidden.



With the Hide/Show Edges cursor, click a vertex near an edge to show or hide the edge

Click again to show the hidden edge.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Reshaping Objects
Editing Vertex-Based Objects

3D Reshape Modes

The **Reshape** tool changes the height and radius of an extruded object, tapered extrude, or a 3D object that is not a mesh object. In addition, use it to reshape walls (see “Editing Walls” on page 535), 3D polygons, and solid primitives (such as cylinders, hemispheres, cones, and spheres), reshape retaining walls (Vectorworks Design Series required), change the angle (rise/run) of roofs created with the **Roof Face** command, and reshape roof objects created with the **Create Roof** command (see “Reshaping Roof Objects” on page 576).

The **Reshape** tool generally has three modes when 3D functionality is enabled. Depending on the selected object, additional modes are available, as described in the associated sections.

For some objects, such as 3D solids, the reshaping functionality is accessed directly from the drawing, and the marquee modes display in the tool bar.

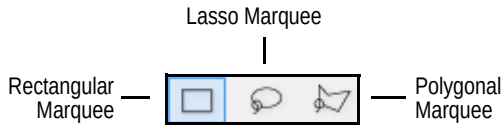
Mode	Description
Move Vertex	Changes the location of a selected vertex or several vertices
Add Vertex	Adds a vertex to the object
Delete Vertex	Deletes a vertex from the object

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Reshaping Extruded Objects and Solid Primitives
Reshaping 3D Polygons
Reshaping NURBS Curves
Reshaping NURBS Surfaces
Reshaping Walls
Reshaping Roof Objects

Marquee Modes

When no objects are selected, the **Reshape** tool has three marquee modes to draw a planar marquee in any view.



Mode	Description
Rectangular Marquee	Creates a rectangular marquee box around several vertices. Click to set the start point, drag the mouse in the desired direction, and release to set the end point. All vertices within the marquee are selected for performing a subsequent reshape operation.
Lasso Marquee	Creates a free-form marquee around several vertices, allowing a more exact selection of vertices of irregular 2D shapes. Click to set the start point, drag the mouse in the desired direction, and release to set the end point. All vertices within the marquee are selected for performing a subsequent reshape operation.
Polygonal Marquee	Creates a marquee with an irregular polygonal shape around several vertices. Click to set the start point, and then continue clicking to define the shape. Double-click to finish the marquee. All vertices within the marquee are selected for performing a subsequent reshape operation.

Click-drag in the drawing to begin drawing a marquee around vertices for reshaping; the currently active marquee mode determines the marquee type that will be drawn. Regardless of what marquee mode is currently active, you can alternatively press and hold the Alt (Windows) or Option (Mac) key to draw a lasso marquee or press and hold Shift+Alt (Windows) or Shift+Option (Mac) to draw a polygonal marquee.

As soon as a reshape marquee exists and an object is selected, the available modes switch to the 2D reshape modes, and 2D reshape functionality is enabled (see “2D Reshape Modes” on page 1044). This allows 2D reshape behavior to be performed, even on a 3D object and in a 3D view, when the vertex is enclosed in the reshape marquee. For example, wall length can be changed with the **Reshape** tool in a 3D view, if the wall end vertex is enclosed within a reshape marquee that is co-planar with the bottom elevation of the wall.

3D objects can be moved if fully inside the marquee and in Top or Top/Plan view.

While using any marquee selection mode, if the vertices to be selected are surrounded by a filled object, first select the object(s) using the **Selection** tool. Press and hold the Shift key while drawing the marquee around the vertices; the currently active marquee mode determines the marquee type that will be drawn. The desired vertices are selected without the boundary object being selected.

Perform the desired reshape operation on the selected vertices.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Performing Multiple Reshapes
The Selection Tool

Performing Multiple Reshapes

Instead of individually reshaping the vertices of objects, one at a time, several vertices and even several objects can be reshaped at one time. In addition, vertices and holes can be deleted from objects. Multiple reshaping can be performed on lines, polylines, walls, dimensions, and polygons. Polygons include all regular and irregular polygons, as well as objects drawn with the **Freehand** tool. For example, use this function to resize walls without moving any of the doors or windows placed inside them, or affecting intersections with other walls.

2D planar objects must be co-planar and in the active plane to be reshaped; the selection marquee is drawn in the active plane. Multiple 3D objects cannot be reshaped at one time, but must be selected individually.

This function does not work on objects drawn with the **Circle**, **Oval**, or **Arc** tools.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

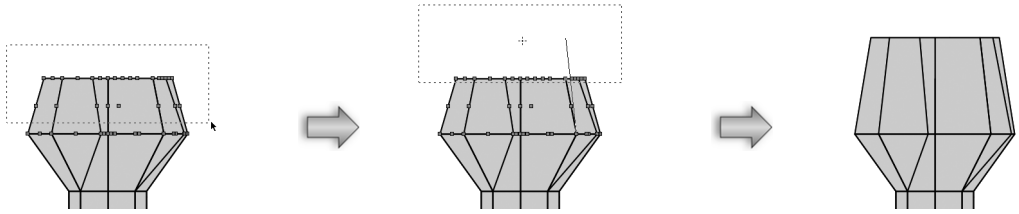
To resize objects with multiple reshape:

1. Select the objects to reshape.
2. Click the **Reshape** tool from the Basic palette, and select **Move Polygon Handles** from the Tool bar.

   The cursor changes to a cross-hair.
3. Click and drag to create a rectangular marquee around multiple vertices of the objects to be resized or deleted.

   To create a lasso marquee, press the Option (Mac) or Alt (Windows) key when creating the marquee. Press the Command and Option (Mac) or Ctrl and Alt (Windows) keys for polygonal marquee selection mode.
4. Change the location of the vertices by either clicking and dragging with the mouse or by using the **Move** command (see “Editing Object Surfaces” on page 1080). Alternatively, nudge the selected vertices by pressing the Shift and arrow keys.

   To use the mouse, click-drag on the vertices and move them to the desired location. A preview displays the current and future location of the object vertices.



Create marquee to select multiple vertices, and then drag selected vertices to new location

Press the Delete key (Mac) or Delete or Backspace keys (Windows) to delete the vertices or holes.

Alternatively, enter an exact length in the Data bar. Press the Tab key while still pressing the mouse button until the **L** field is highlighted, enter a value for the length, and release the mouse button.

Walls are resized with the wall intersections, doors, or other symbols maintained in place. The marquee remains visible until another tool or command is selected.

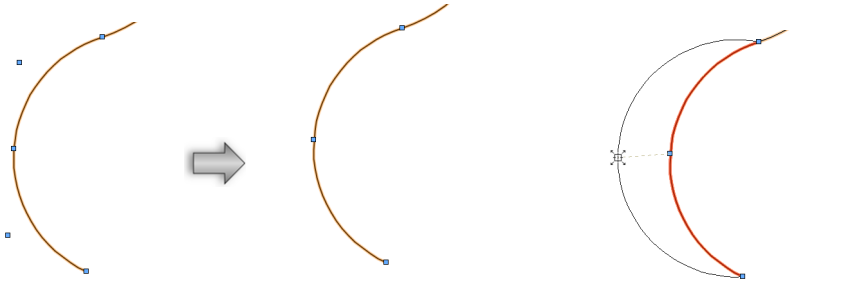
Reshaping Objects
Converting Polyline Vertices from Arc to Radius
Reshaping Polylines

Converting Polyline Vertices from Arc to Radius

As of Vectorworks version 2012, a new type of polyline vertex, the radius vertex, simplifies the reshaping of polylines. Polylines from older versions may include arc vertices, which are harder to edit. Each vertex of a polyline can be changed to radius type with the Change Vertex mode of the **Reshape** tool, or by editing the vertex type in the Object Info palette. However, this is tedious if the object includes several arc vertices. The **Convert Arc to Radius Polyline** command is available to convert all of the arc vertices to radius vertices. Objects like hardscapes or slabs (Vectorworks Design Series required), which are based on path polylines, can benefit by converting their path polyline to contain radius vertices, because they become easier to reshape.

To convert arc vertices to radius vertices:

1. Select the polyline; for path-based objects, double-click on the object or select **Modify > Edit [Object]** to access the path in editing mode. See “Object Editing Mode” on page 1004.
2. Right-click (Windows) or Ctrl-click (Mac) on a polyline which contains at least one arc vertex, and select **Convert Arc to Radius Polyline** from the context menu.
3. The arc vertices of the polyline or path object are converted to radius vertices.



Radius vertices are easier to edit with the Reshape tool

Reshaping Objects
Editing Vertex-Based Objects
Reshaping Polylines
Change Vertex Mode
Smoothing Objects

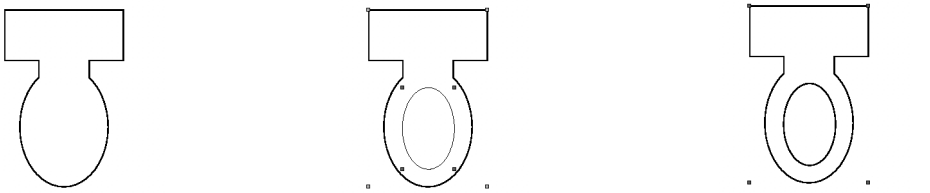
Reshaping Polylines

Polylines can be reshaped in a manner that is similar to an Edit Group operation. The polyline itself is locked in this process, but holes can be added to it, filleted, chamfered, offset, duplicated, extracted, and deleted from it.

To reshape a polyline:

1. Select the polyline to change.
2. Select **Modify > Edit Polyline**.
3. Select polyline holes and move, delete, or reshape them. Add new holes to the polyline, or move a hole out of the polyline to extract it, creating individual polylines.

   The original polyline is locked and cannot be edited.
4. Click **Exit Polyline** to return to the drawing.



Original polyline | In Edit Polyline mode, a hole is added | The polyline has been edited

Reshaping Objects
Editing Vertex-Based Objects
Smoothing Objects
Converting Polyline Vertices from Arc to Radius

Reshaping Extruded Objects and Solid Primitives

The **Reshape** tool changes the height, and when applicable, the radius of extrudes, cylinders, spheres, hemispheres, and cones, and the height and angle of tapered extrudes.

To reshape an extrude, cylinder, sphere, hemisphere, cone, or tapered extrude:

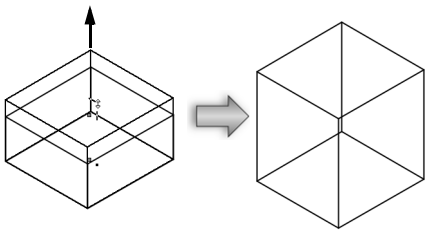
1. Select the object to reshape.
2. Click the **Reshape** tool from the Basic palette.

   A resize handle is added to both top and bottom of an extruded object. Multiple resize handles are added around cylinder, sphere, hemisphere, and cone objects. For a tapered extrude object, one resize handle is added to its top for changing its height, and one to the side for changing its taper angle.

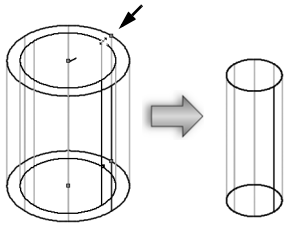
   Tapered extrudes with a taper angle of zero have an additional resize angle at the bottom.
3. Click and drag the resize handle to change the object's height, radius, or taper angle.

   When the cursor is over a radius resize handle, the standard arrow cursor changes into a double-headed arrow. It changes to an unfilled double-headed arrow over a height resize handle.
4. Click again when the object is at the desired height, radius, or angle.

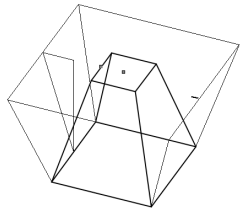
   Alternatively, press the Tab key and enter a numeric value for the height, radius, or angle in the Data bar.



With the Reshape cursor, drag the resize handle up (or down) to the desired extrude height



Change the radius of a cylinder, sphere, hemisphere, or cone by dragging the resize handle to the desired radius



Change the angle of a tapered extrude by dragging the side resize handle to the desired angle

Alternatively, use the **Push/Pull** tool for these operations. Changing the top or bottom face of an extrude does not change the object type; it remains an extrude.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Reshaping Objects
Direct Modeling with the Push/Pull Tool

Reshaping 3D Polygons

The **Reshape** tool, located in the Basic palette, can manipulate and reshape 3D polygons.

To reshape 3D polygons:

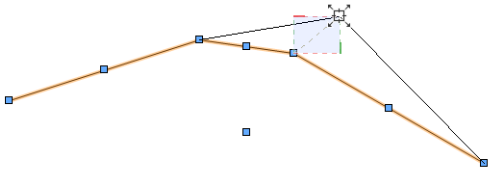
1. Select the 3D polygon to reshape. Only one 3D polygon can be selected for reshaping at a time.
2. Click the **Reshape** tool from the Basic palette.

   Select the mode from the Tool bar.

Mode	Description
Move Vertex	Changes the location of a selected vertex or several vertices
Add Vertex	Adds a vertex to the object
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mode	Description
Delete Vertex	Deletes a vertex from the object

3. Select a vertex. When the cursor is over a vertex, the standard arrow cursor changes to an unfilled, four-way arrow.



4. Reshape the 3D polygon.

- In **Move Vertex** mode, click-drag the mouse to move the vertex or vertices. The selected constraint mode restricts the movement to the working plane or to one of the axes. Release the mouse at the desired location.

  Alternatively, enter the exact coordinates in the Data bar. Press the Tab key, enter the coordinates, and then press Enter.

- To add a vertex in **Add Vertex** mode, move the cursor to an existing vertex near the location where the new vertex is to be added. The cursor becomes an arrow with two black diamonds when a vertex can be added. Drag the cursor to the desired location for the new vertex. Click at the new location.
- To delete a vertex in **Delete Vertex** mode, click on the vertex to delete.

Functionality similar to the **Reshape** tool is available from the Object Info palette Shape tab; see “Editing Vertex-Based Objects” on page 1002.

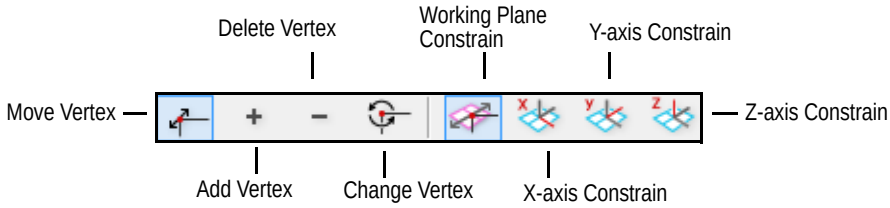
Reshaping NURBS Curves

The **Reshape** tool, located in the Basic palette, can manipulate and reshape NURBS curves.

To reshape NURBS curves:

1. Select the NURBS curve to reshape. To reshape multiple vertices, the curve must be a single, smooth continuous NURBS curve.
2. Click the **Reshape** tool from the Basic palette.

   Select the mode from the Tool bar.

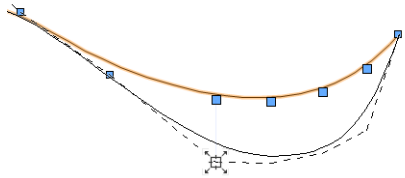


Mode	Description
Move Vertex	Changes the location of a vertex or several vertices
Add Vertex	Adds a vertex to the NURBS curve
Delete Vertex	Deletes a vertex from the NURBS curve
Change Vertex	Toggles an existing vertex between a corner vertex and smooth vertex

Mode	Description
Working Plane Constrain	Moves the selected vertex or vertices on a working plane
X-axis constrain	Moves the selected vertex or vertices along the X axis
Y-axis constrain	Moves the selected vertex or vertices along the Y axis
Z-axis constrain	Moves the selected vertex or vertices along the Z axis

3. Select a vertex. When the cursor is over a vertex, the standard arrow cursor changes to an unfilled, four-way arrow.

   Alternatively, to select several vertices at one time in **Move Vertex** mode, click with the Shift key pressed or click and drag to create a marquee around the desired vertices. Position the cursor over one of the vertices.



4. Reshape the NURBS curve.

- In **Move Vertex** mode, click-drag the mouse to move the vertex or vertices. The selected constraint mode restricts the movement to the working plane or to one of the axes. Release the mouse at the desired location.

  Alternatively, enter the exact coordinates in the Data bar. Press the Tab key, enter the coordinates, and then press Enter.

- To add a vertex in **Add Vertex** mode, move the cursor to an existing vertex near the location where the new vertex is to be added. The cursor becomes an arrow with two black diamonds when a vertex can be added. Drag the cursor to the desired location for the new vertex. Click at the new location.
- To delete a vertex in **Delete Vertex** mode, click on the vertex to delete.
- Toggle the vertex between a corner vertex and a smooth vertex by clicking on the vertex in **Change Vertex** mode.

Functionality similar to the **Reshape** tool is available from the Object Info palette Shape tab; see “Editing Vertex-Based Objects” on page 1002.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating NURBS Curves
Reshaping Objects

Reshaping NURBS Surfaces

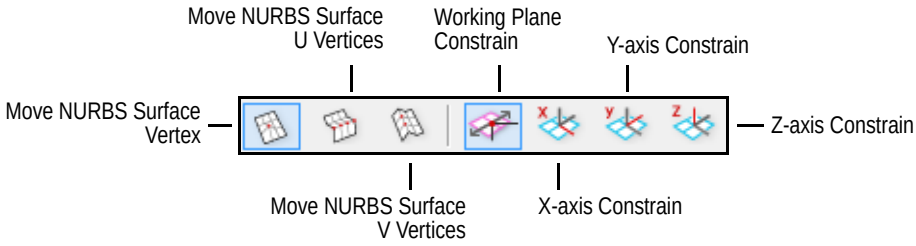
The **Reshape** tool, located in the Basic palette, can manipulate NURBS surface control points and to reshape NURBS surfaces. For example, a dome or bell-curve effect can be created by manipulating the vertices.

This tool can potentially create surfaces which cannot be further manipulated in the 3D Power Pack. See “Surface Geometry Requirements” on page 327.

To reshape a NURBS surface:

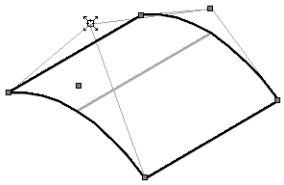
1. Select the NURBS surface to reshape.
2. Click the **Reshape** tool from the Basic palette.

   Select the mode from the Tool bar.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Mode	Description
Move NURBS Surface Vertex	Changes the position of the selected vertex or vertices only
Move NURBS Surface U Vertices	Changes the position of all vertices in a row in the U direction
Move NURBS Surface V Vertices	Changes the position of all vertices in a row in the V direction
Working Plane Constrain	Moves the selected vertex or vertices on a working plane
X-axis Constrain	Moves the selected vertex or vertices along the X axis
Y-axis Constrain	Moves the selected vertex or vertices along the Y axis
Z-axis Constrain	Moves the selected vertex or vertices along the Z axis

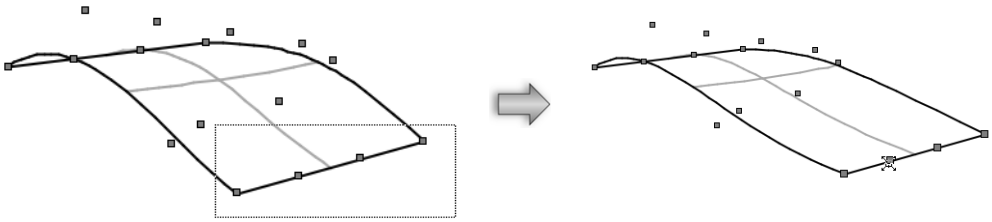
To select one vertex or row of vertices, position the cursor over one of the NURBS surface vertices. When the cursor is over a vertex, the standard arrow cursor changes to an unfilled, four-way arrow.



To select several vertices at one time, click with the Shift key pressed or click and drag to create a marquee around the desired vertices.

If several vertices are selected, position the cursor over one of them.

3. Click-drag the mouse to move the vertex or vertices.
4. Release the mouse at the desired location.



Functionality similar to the **Reshape** tool is available from the Object Info palette Shape tab; see “Editing Vertex-Based Objects” on page 1002.

NURBS Surfaces
Reshaping Objects
Direct Modeling with the Push/Pull Tool

Offsetting Objects

The **Offset** tool either creates a duplicate object offset from the original, or offsets the selected object from its original location. Use this tool to easily create parallel objects, such as lines and walls. Also use it to produce a larger or smaller version of closed objects, such as ovals and connected walls. NURBS surfaces are offset by the offset distance along the surface normal direction.

The Offset tool can be used with the following objects.

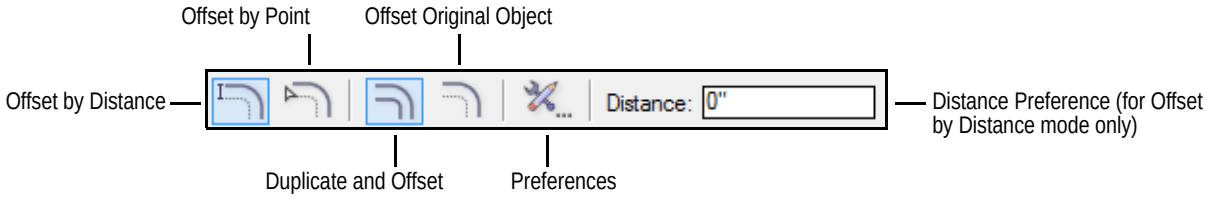
Vectorworks Fundamentals Objects

- NURBS curves
- Walls
- Open 2D objects (arcs, lines, polylines)
- NURBS surfaces
- Revision clouds
- Closed 2D objects (circles, ovals, rectangles, rounded rectangles, polylines, polygons)

Vectorworks Design Series Objects

- Hardscapes
- Seating layouts
- Stipples
- Massing models
- Site modifiers
- Plants
- Property lines
- Spaces
- Redlines

The **Offset** tool has four modes, which are also options in the Offset Tool Preferences dialog box.



The **2D conversion resolution** field, on the Edit tab of the Vectorworks Preferences dialog box, adjusts the degree of smoothing. The higher the conversion resolution, the higher the degree of smoothing, which produces a more accurate offset of objects. For more information on 2D conversion resolution, refer to “Edit Preferences” on page 49.

To offset one or more objects:

1. Select the object(s) to be offset, if desired.
2. Click the **Offset** tool from the Basic palette.
3. Click **Preferences** from the Tool bar.

   The Offset Tool Preferences dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Method	
Offset by Distance	Places the offset object at the specified **Distance** from the original location (same as selecting the Tool bar button and entering the distance on the Tool bar)

Parameter	Description
Offset by Point	Places the offset object at a distance specified by a mouse click (same as selecting the Tool bar button)
Duplication	
Duplicate and Offset	Creates a duplicate of the object at the offset location (same as selecting the Tool bar button)
Offset Original Object	Moves the original object to the offset location (same as selecting the Tool bar button)
Wall Offset	
Offset from Center Line	Offsets walls from the center line of the wall's original location
Offset from Nearest Edge	Offsets walls from the nearest edge of the wall's original location
Smooth Corners	Rounds sharp corners in the offset object
Close Open Curves	Draws lines at both ends of the offset and original objects, to create a closed shape from the open curves

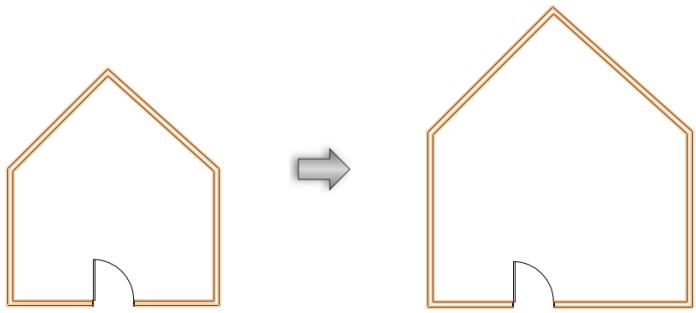
4. Specify the desired preferences and click **OK**.

   To select additional objects or to change the currently selected objects to offset (or duplicate and offset), press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd key is being pressed.

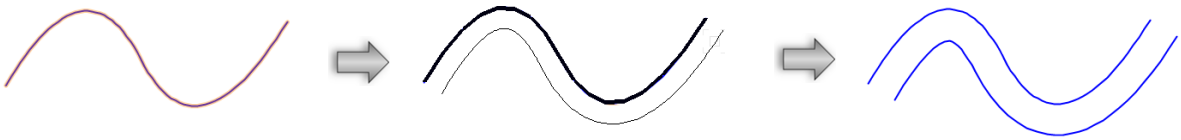
5. Depending on the offset preferences, do one of the following:

Offset Method	Description
Offset by Distance	
Click on objects to offset	Select the object to be offset; the object is highlighted. Click to specify the offset location relative to the selected object, and to place the offset object. To continue offsetting objects, click another object to highlight it, and then click again to indicate the offset location.
Offset selected objects	Click in the drawing to specify the offset location relative to the originally selected object, and to place the offset object. The offset object is now selected; to offset from that object, click again to indicate the offset location.
Offset by Point	
Click on objects to offset	Select the object to be offset; the object is highlighted. Move the mouse to adjust the offset preview if necessary, and then click to place the offset object. To continue offsetting objects, click another object to highlight it, and then click again to indicate the offset location.
Offset selected objects	Click to set the offset point (move the mouse to adjust the offset preview if necessary), and then click again to place the offset object. The offset object is now selected; to offset from that object, click again to indicate the offset location.

Depending on the offset preferences, either a duplicate object or the original object is placed at the offset location.



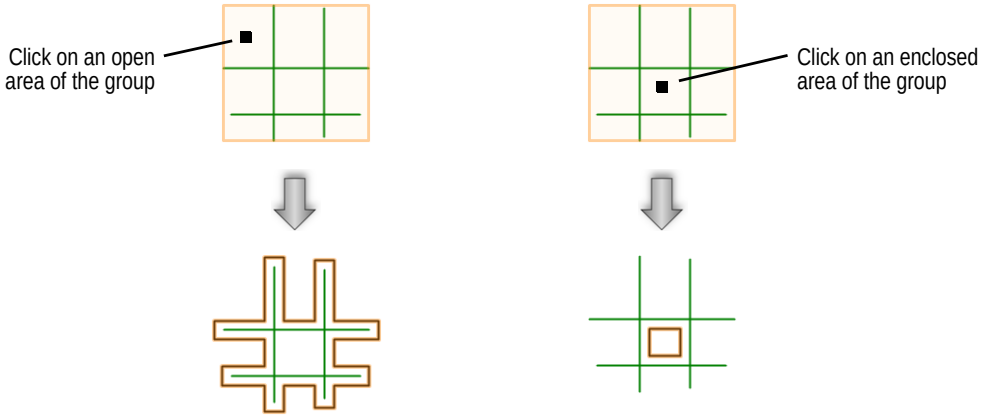
Select the walls with the Selection tool, and then select the Offset tool. Select Offset by Distance mode, and set the offset Distance to 2'. Click outside the walls to offset them 2' beyond the original location.



In Offset by Point mode, click the object to be copied and offset; the object is highlighted

Click the point where the offset is to be placed; a preview displays

Click again to place the offset object



For grouped objects, the offset result depends on where you click

Click here for a video tip about this topic (Internet access required).

Trimming and Clipping Objects

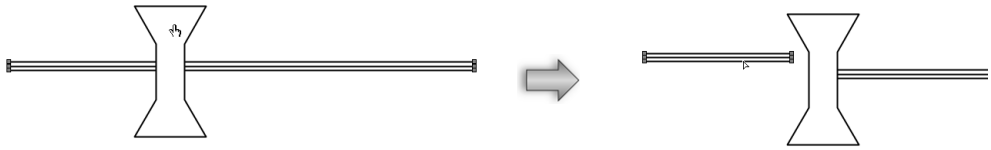
Trim Command

Use the **Trim** command to trim lines or walls where they intersect with another object.

To trim objects:

1. If necessary, position the trimming object over the object(s) to be trimmed.
2. Select the trimming object.
3. Select **Modify > Trim**.
4. Click on the object to be trimmed.

   The trim command is executed and the pieces can be moved independently.



Trim Tool
Clip Tool

Trim Tool

The **Trim** tool trims a portion of the object indicated. 2D objects that can be edited in this manner include lines, arcs, rectangles, rounded rectangles, circles, ovals, polylines, and polygons. Straight walls can also be trimmed to intersecting straight walls using the **Trim** tool. The walls are automatically joined, regardless of whether the Vectorworks preference **Auto join walls** is on.

The **Trim** tool has two modes that determine how the object is trimmed.

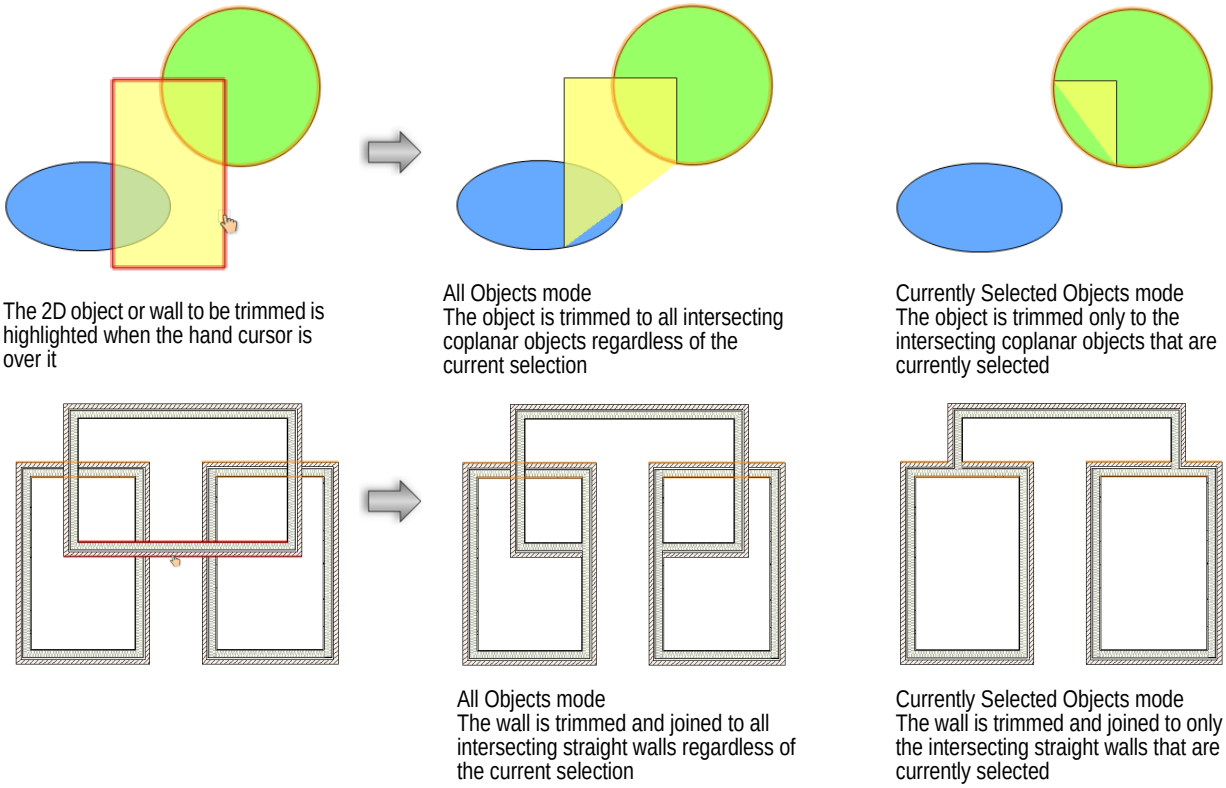
Mode	Description
All Objects	The object is trimmed to all intersecting coplanar objects/straight walls regardless of the current selection
Currently Selected Objects	The object is trimmed only to the intersecting coplanar object(s)/straight wall(s) that is currently selected

To trim a portion of an object:

1. Click the **Trim** tool from the Basic palette.

   The standard arrow cursor changes into a hand.
2. Click the **All Objects** or **Currently Selected Objects** mode from the Tool bar.
3. Position the hand cursor over the object to be trimmed; the object to be trimmed is highlighted. Only one object can be trimmed at a time.
4. Click the portion of the object to be trimmed.

   The object is trimmed to intersecting coplanar objects/straight walls according to the mode selected.



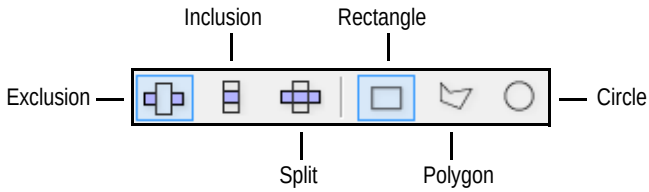
To trim another portion of the same object, move the hand cursor to the desired location and click.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Trim Command
Clip Tool

Clip Tool

The **Clip** tool cuts out pieces from 2D objects, such as lines, arcs, rectangles, rounded rectangles, circles, ovals, polygons, and polylines. In addition, this tool can be used to split an object. The **Clip** tool operates on all selected objects in the active plane.



Mode	Description
Exclusion	Cuts a hole in the object(s) according to the clipping object shape
Inclusion	Trims everything away from the outside of the clipping object shape
Split	Splits the object(s) and the clipping area into separate objects
Rectangle, Polygon, Circle	Each mode can use one of the clipping shapes; select rectangle, polygon, or circle

To clip objects:

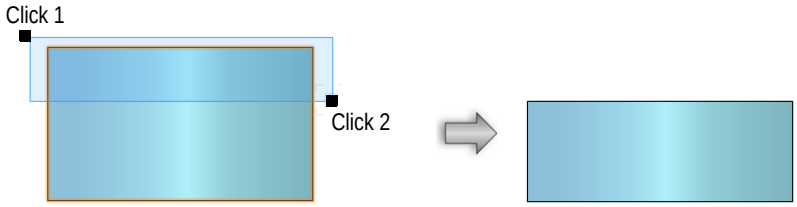
1. Select the object or objects to cut.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Click the **Clip** tool from the Basic palette.

   To select additional objects or to change the currently selected objects to clip, press and hold the Alt key (Windows) or Cmd key (Mac) while shift-clicking or click-dragging the mouse around them to marquee select them; the **Selection** tool is activated in boomerang mode while the Alt or Cmd key is being pressed.

3. Select the clipping mode and the clipping object shape.
4. Click and drag to create a marquee box. The marquee is defined on the active plane.

   The object is clipped as defined by the clipping object shape.



Click here for a video tip on this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Trim Command
Trim Tool
Clip Surface

Resizing Objects

Fixed Point Resize Tool

The **Fixed Point Resize** tool resizes rectangles, rounded rectangles, polygons, circles, and ovals using a fixed point on the drawing as the point of reference. It is best used to scale an object relative to a particular location in the drawing. To rescale an object symmetrically or asymmetrically by a specific factor, use the **Scale Objects** command.

To resize an object:

1. Select the object or objects to change.
2. Click the **Fixed Point Resize** tool from the Basic palette.
3. Click on the point in the drawing to serve as the fulcrum.

   To scale the resized object symmetrically, select a fulcrum point that is at the exact center of the object.

   If the fixed point is not at the center of the object, the resized object changes proportions.

4. Click on an object edge and drag the object into its new size, shape, and/or location.

If the drag point is moved past the selected fulcrum, the object is reversed.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

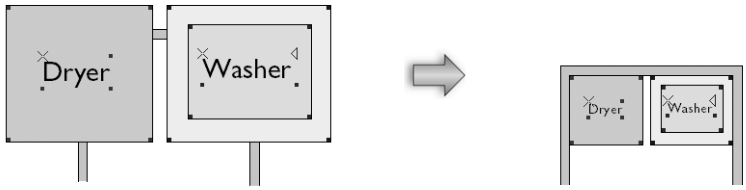
The starting point for dragging cannot be the same as the fulcrum point or exactly horizontal or vertical to it.

Scaling Objects

Scaling Objects

The **Scale Objects** command rescales the X and/or Y dimension of any selected solid, object, or group, or rescales the selected items uniformly in the X,Y, and Z dimensions. The selected item is rescaled using its center point. The object can be rescaled by indicating a segment on the drawing and entering a new distance for the segment. If no objects are selected, the entire drawing can be scaled. Layer links cannot be scaled with this command.

Unless you are scaling the entire drawing, the **Scale Objects** command does not scale symbols directly, although it can scale symbols in object editing mode. Use the Object Info palette or the **Selection** tool to scale a symbol instance directly in the drawing, as described in "Scaling Symbols from the Object Info Palette" on page 1064.



To scale objects:

1. Select the object(s) to scale.

   If no objects are selected, all objects on all layers of the drawing are scaled.

2. Select **Modify > Scale Objects**.

   The Scale Objects dialog box opens. Specify the scaling method and parameters.

Click to show/hide the parameters.

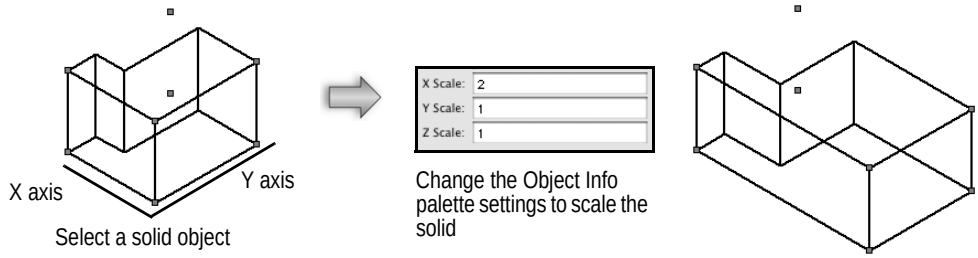
Parameter	Description
Symmetric	Scales uniformly along X, Y, and Z axes
X, Y, Z Factor	Enter the scaling factor (for example, enter 2 to double or .5 to halve the scale)
Symmetric By Distance	Scales symmetrically using the ratio of the current and new distance values as a scale factor
Current Distance	Enter the current distance to be scaled, or click the button to use a temporary tool to indicate the distance on the drawing
New Distance	Enter the new value for the distance
Asymmetric	Scales along only the specified X and Y axes; when asymmetrically scaling a solid, the current view must be aligned with the solid's matrix for scaling to occur
X / Y Scaling Factor	Enter the scaling factor (for example, enter 2 to double or .5 to halve the scale)
Scale text	Scales selected text to the new scaling factor
Entire drawing	Scales all objects on all layers, including text, symbols, and hatches

3. If specifying the scale factor with the **Symmetric by Distance** option, click the button to switch temporarily to the drawing. Click to indicate the start of the segment, move the mouse, and click to indicate the end of the segment. You are returned to the Scale Objects dialog box, and the **Current Distance** value has been specified. Enter the **New Distance** for the segment.
4. Click **OK**.

Scaling Solids Asymmetrically

Solids can be scaled asymmetrically, through both the **Scale Objects** command (see "Scaling Objects" on page 1063) and the Object Info palette. The internal components of the solid do not change, and the solid can still be edited after the scaling operation. Spheres, hemispheres, and cones cannot be scaled asymmetrically.

In the Object Info palette, enter a scale factor in the **X Scale**, **Y Scale**, or **Z Scale** field to scale the selected solid along the specified axis.



A selected solid can also be scaled asymmetrically by selecting the **Modify > Scale Objects** command and entering an Asymmetric scale factor. However, the current view must be aligned with the solid's matrix for scaling to occur.

Scaling Symbols from the Object Info Palette

Symbol instances placed in the drawing can be scaled from the Object Info palette; this is an advantage because different sizes of the same symbol do not require a new symbol for each size.

> Symbols can also be scaled interactively with the interactive scaling mode of the **Selection** tool. See "The Selection Tool" on page 109.

To scale a symbol instance:

1. Select the symbol instance in the drawing area.
2. From the Object Info palette, select Symmetric or Asymmetric **Scaling**.
3. For Symmetric scaling, enter the scale **Factor**. For Asymmetric scaling, enter the scaling factor along the specified axis.

   Page-based symbols can only be scaled symmetrically.
4. The symbol is resized. The Object Info palette displays **Scaled** at the top of the Shape pane for symbols that have been scaled. Other instances of the same symbol in the drawing remain unscaled.

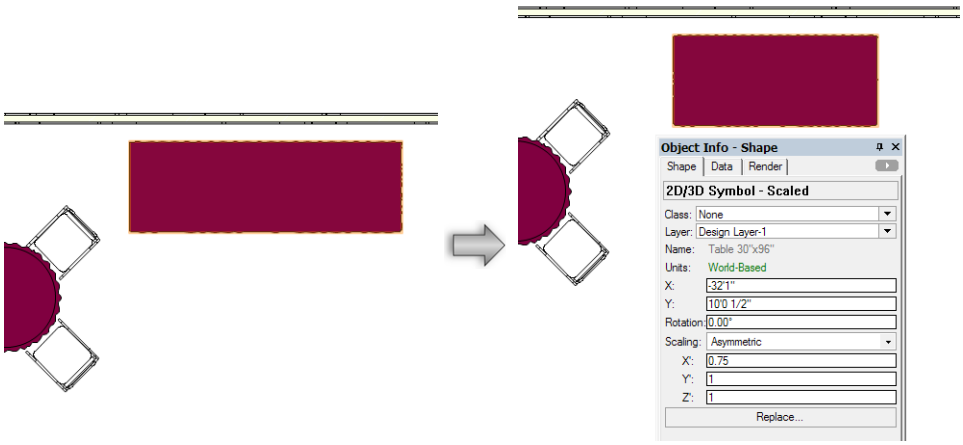


Table is too long and too close to chair

Table scaled along X axis

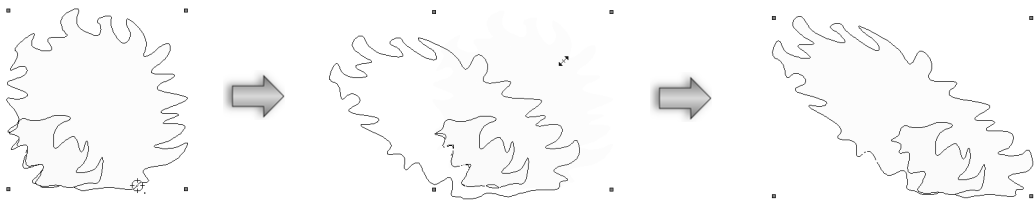
Object Editing Mode

Shearing Objects

The **Shear** tool simultaneously skews all of the vertices of a rectangle, rounded rectangle, polygon, circle, oval, or polyline, using a fixed point on the drawing as a point of reference. It is like placing a thumbtack on one point of an object, or the drawing, and reshaping every other object point except the one tacked down. The **Shear** tool can be used to fake a 3D perspective.

To shear an object:

1. Select the object or objects to change.
2. Click the **Shear** tool from the Basic palette.
3. Click on the point in the drawing to tack down.
4. Click on the object and drag the resize cursor to shear the object. A preview object displays.
5. Click to set the shear position.



Click to tack down a point then drag the resize cursor to shear the object

Click here for a video tip on this topic (Internet connection required).

Joining Objects

Join Command

The **Join** command can be used to join two single lines, two double lines, or two walls together. Joined lines intersect but remain as individual objects.

Object	Command	Specifications
Individual Lines	**Join** **Join (No Trim)**	An individual line can be joined to another individual line
Double Lines	**Join** **Join and Fillet**	Double lines can be joined to another set of double lines or to a wall. Double lines drawn with the **Create Polygons** option (set in Double Line preferences) cannot be joined. Only the line elements of double lines drawn with the **Create Lines and Polygons** option can be joined.
Walls	**Join** **Join (No Trim)**	A wall can be joined to another wall or to double lines created with the **Create Lines** option (set in Double Line preferences). Walls will join to the line elements of double lines drawn with the **Create Lines and Polygons** option.

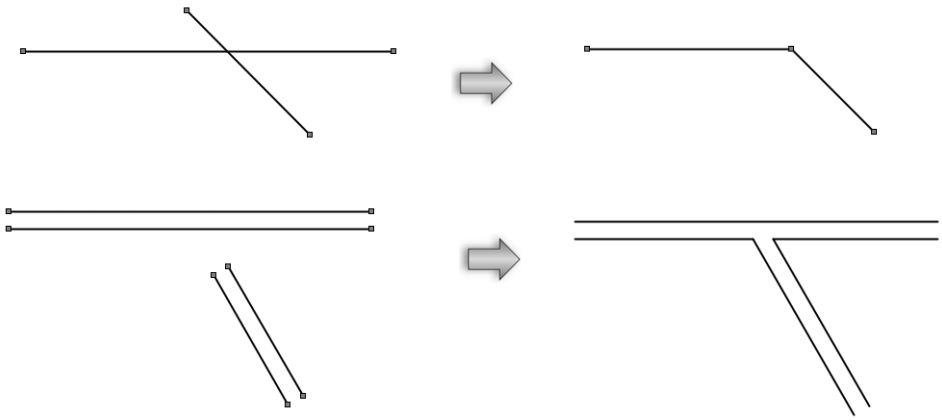
Join
Join and Fillet
Join (No Trim)

Join

To join walls, single lines, or double lines:

1. Select the two non-parallel walls, lines, or double lines to join.
2. Select **Modify > Join > Join**.

   The selected walls/lines are joined together with any excess trimmed away.



Join and Fillet

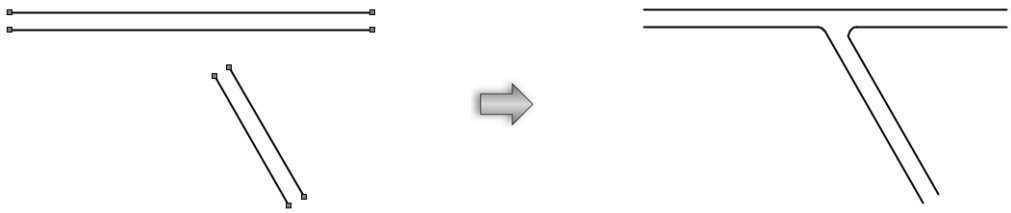
To join and fillet double lines:

1. Select the two sets of non-parallel double lines to join.
2. Select **Modify > Join > Join and Fillet**.

   If a fillet measurement has already been specified for this drawing, the two double lines are connected and filleted using the default value.

   If a fillet measurement has not yet been specified for this drawing, the Fillet Settings dialog box opens.

   Enter the desired fillet radius and click **OK** to connect and fillet the lines.



The Fillet Settings dialog box is accessible from various locations in the Vectorworks program, whenever a fillet radius value can be set. A change to the radius value in one location changes the default setting in all locations.

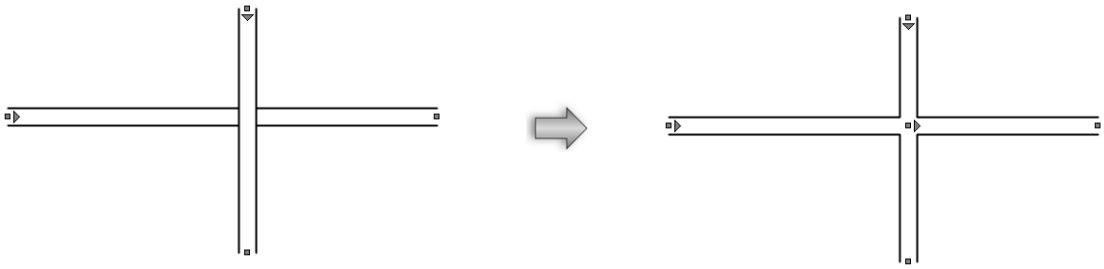
Fillet Tool

Join (No Trim)

To join two overlapping walls or individual lines, without trimming away the excess:

1. Select the two non-parallel walls or lines to join.
2. Select **Modify > Join > Join (no trim)**.

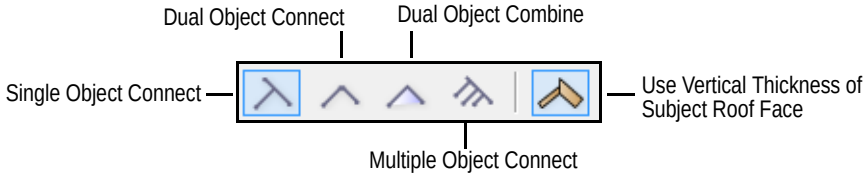
   The selected walls/lines are joined without trimming any excess.



Combining and Connecting Objects

Connect/Combine Tool

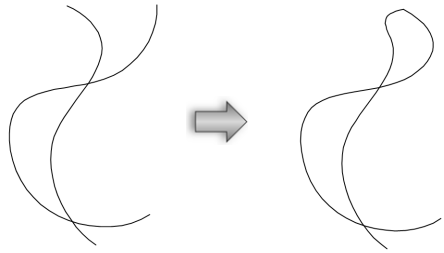
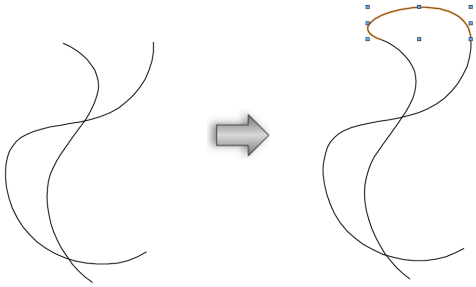
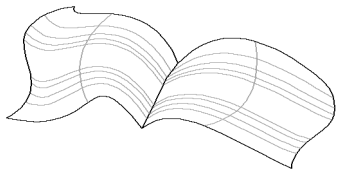
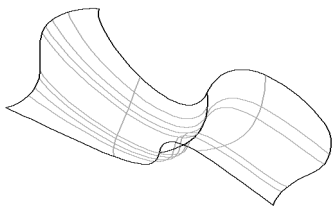
The **Connect/Combine** tool joins objects by their endpoints or at their intersections using one of four modes.

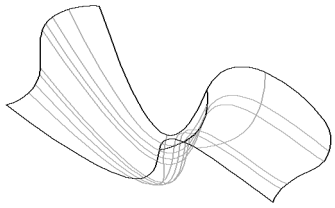


Mode	Description
Single Object Connect	Trims or extends the first selected object to join a second or boundary object This mode can be used to connect roof faces.
Dual Object Connect	Trims or extends to join two objects at their endpoints or intersections This mode can be used to connect roof faces.
Dual Object Combine	Trims or extends to join two objects into one object at their endpoints or intersections
Multiple Object Connect	Designates the first selected object as a boundary, and then trims or extends multiple consecutively selected objects to join the boundary

Mode	Description
Use Vertical Thickness of Subject Roof Face	If the tool is being used to connect two roof faces that have different thicknesses, this option automatically adjusts the roof face thickness of the second clicked roof face to match the vertical thickness of the first clicked roof face.

When connecting or combining a pair of NURBS, polylines, or open polygons by their endpoints, you may need to supply additional information to complete the process. In this situation, the Connect/Combine Options dialog box opens.

Option	Description
Mid-Point	Connects/combines the two objects by joining the two endpoints midway between them 
Blend	Connects/combines the two objects by creating another object between them 
Position Matching (NURBS curves and surfaces only)	Connects/combines two NURBS curves or surfaces by moving the first object's selected end point to the selected end point of the boundary object (not available if the end points are coincident) 
Tangency Matching (NURBS curves and surfaces only)	Connects/combines two NURBS curves or surfaces by making the first object's selected end point tangent to the selected end point of the boundary object 

Option	Description
Curvature Matching (NURBS curves and surfaces only)	Connects/combines two NURBS curves or surfaces by making the first object's selected end point match the curvature of the selected end point of the boundary object 

Click here for a video tip on this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Single Object Connect

The **Single Object Connect** mode trims or extends one or more selected objects to join a boundary object. Only open objects, such as lines, arcs, open polygons, and NURBS curves and lines, can be connected. Closed objects, such as circles, rectangles, and closed polygons cannot be connected; they are treated as boundary objects.

This mode can also be used to connect roof faces; see "Connecting Roof Faces" on page 569.

Connecting Single Objects

To connect a single object to a boundary object:

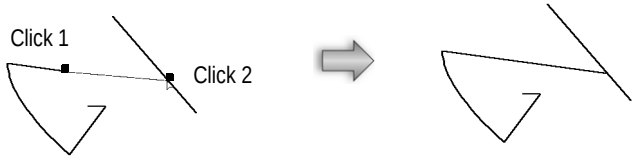
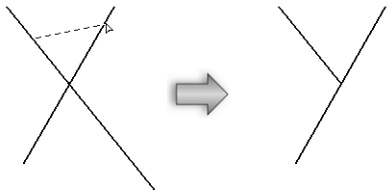
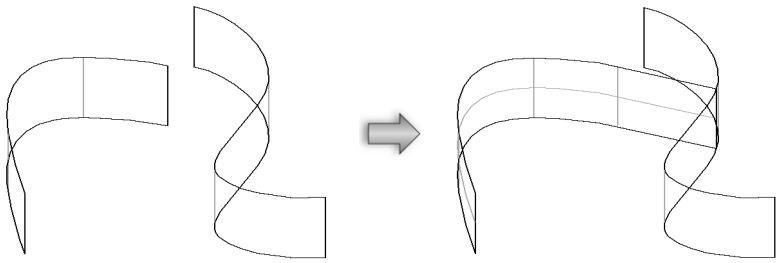
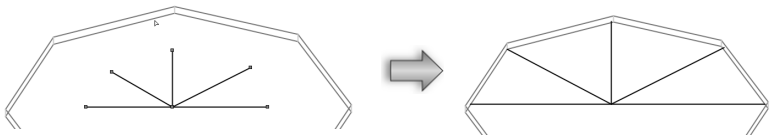
1. Click the **Connect/Combine** tool from the Basic palette, and select **Single Object Connect** from the Tool bar.
2. Click on the object to connect, and then click on the boundary object. The first object is resized to join the boundary object.

Connecting Multiple Objects

To connect multiple selected objects to a boundary object:

1. Select the objects to be connected to a boundary object.
2. Click the **Connect/Combine** tool from the Basic palette, and select **Single Object Connect** from the Tool bar.
3. While pressing and holding Alt (Windows) or Option (Mac), click on one of the objects to connect, and then click on the boundary object. The selected objects are resized to join the boundary object.

   Alternatively, use Multiple Object Connect mode to connect multiple objects to a boundary object (see "Multiple Object Connect" on page 1072).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Connection Type	Example
2D object extended to boundary object	
2D object trimmed at boundary object	
NURBS surface to NURBS surface	
Multiple selection connecting 2D objects to boundary object	

Connect/Combine Tool

Dual Object Connect

The **Dual Object Connect** mode trims or extends two objects to connect them at their endpoints or intersections. Only open objects, such as lines and polylines, can be connected. Closed objects, except for NURBS surfaces, cannot be connected.

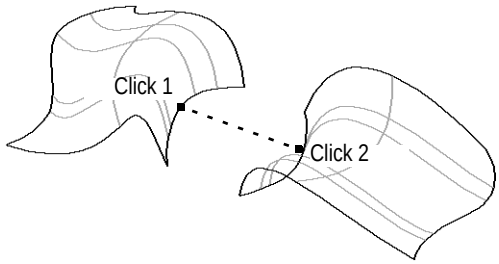
This mode can also be used to connect roof faces; see “Connecting Roof Faces” on page 569.

To connect two objects:

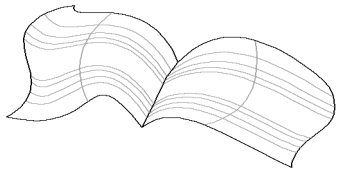
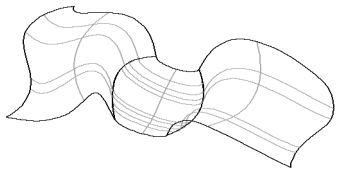
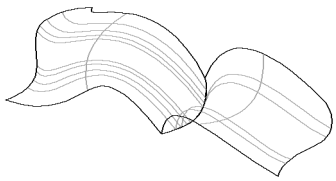
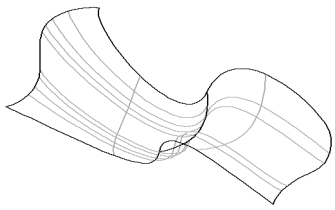
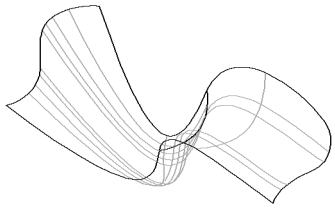
1. Click the **Connect/Combine** tool from the Basic palette, and select **Dual Object Connect** from the Tool bar.
2. Click the first, and then the second, object to connect.

   The Connect/Combine Options dialog box opens. Select a method for resizing and connecting the two objects.

   The two objects are resized to connect to each other.



NURBS surface being connected to NURBS surface

Connection Type	Example
Mid-Point	
Blend	
Position Matching	
Tangency Matching	
Curvature Matching	

Connect/Combine Tool

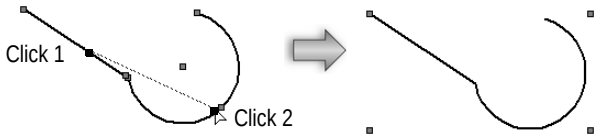
Dual Object Combine

The **Dual Object Combine** mode trims or extends to combine two objects into a single object at their endpoints or intersections. Only open objects, such as lines and NURBS curves and surfaces, can be connected.

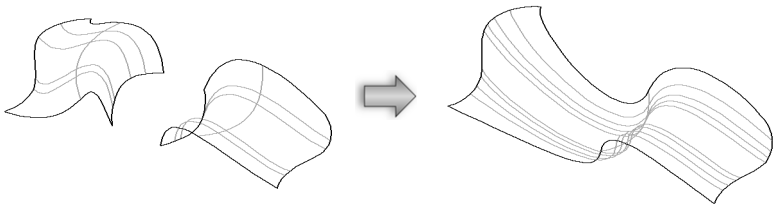
To combine two objects:

1. Click the **Connect/Combine** tool from the Basic palette, and select **Dual Object Combine** from the Tool bar.
2. Click on the first, and then the second, object to combine.

   The two objects are trimmed or extended if necessary and combined into one object.



The line and the arc are combined to form a single polyline object



Tangent or curvature matching between two NURBS surfaces creates a single surface

Connect/Combine Tool

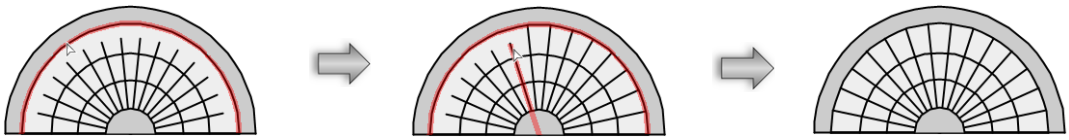
Multiple Object Connect

The **Multiple Object Connect** mode trims or extends multiple objects to join a selected boundary object. Only open objects, such as lines, arcs, open polygons, and NURBS curves and lines, can be connected. Closed objects, such as circles, rectangles, and closed polygons can only be used as boundary objects.

Alternatively, use the Alt (Windows) or Option (Mac) key in Single Object Connect mode to connect multiple objects to a boundary object (see “Single Object Connect” on page 1069).

To connect multiple objects to a boundary object:

1. Click the **Connect/Combine** tool from the Basic palette, and select **Multiple Object Connect** from the Tool bar.
2. Click on the boundary object, which becomes highlighted in red.
3. Position the cursor over the first object to trim or extend to the boundary; it becomes highlighted in red. Click to resize the object to join the boundary object.
4. Continue to click on objects to connect to the boundary. Each consecutively clicked object is connected, until you click in an empty space to deselect the boundary object.



The first click selects the boundary object

As each line is clicked, it is connected to the boundary object

Connect/Combine Tool

Splitting Objects and NURBS Surfaces with the Split Tool

The **Split** tool splits the following objects.

Vectorworks Fundamentals Objects

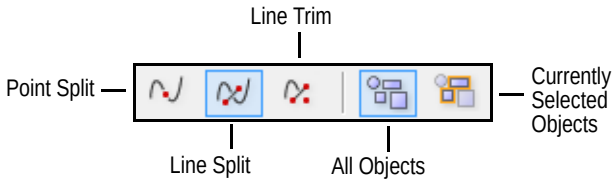
- 3D solid objects (extrudes, cones, cylinders, spheres, hemispheres)
- Revision clouds
- NURBS curves
- NURBS surfaces
- Viewports
- 2D objects (ovals, circles, rectangles, rounded rectangles, arcs, lines, polylines, polygons)
- Pillars
- Walls
- Floors
- Roof faces

Vectorworks Design Series Objects

- Property lines
- Massing models
- Site modifiers
- Seating layouts
- Redlines
- Stipples
- Spaces
- Hardscapes

Splitting an object may change its type; for example, splitting a roof face creates a solid section, which can no longer be edited as a roof face.

The following modes are available for the **Split** tool.



Mode	Description
Point Split	Cuts an object or NURBS surface at a specified point
Line Split	Splits an object or NURBS surface along a cutting line
Line Trim	Splits an object or NURBS surface along a cutting line, and then keeps a designated side
All Objects	For **Line Split** and **Line Trim** modes, splits all objects along a cutting line regardless of the current selection
Currently Selected Objects	For **Line Split** and **Line Trim** modes, splits only the currently selected objects along a cutting line

Surfaces generated by successive splitting can be joined together with the **Compose** command (see “Composing and Decomposing Objects and Surfaces” on page 1020).

This tool may not be able to manipulate certain types of surface geometry (see “Surface Geometry Requirements” on page 327).

Click here for a video tip on this topic (Internet connection required).

Point Split Mode

Line Split Mode
Line Trim Mode
Splitting or Trimming NURBS Surfaces

Point Split Mode

The **Point Split** mode cuts an object at a specified location.

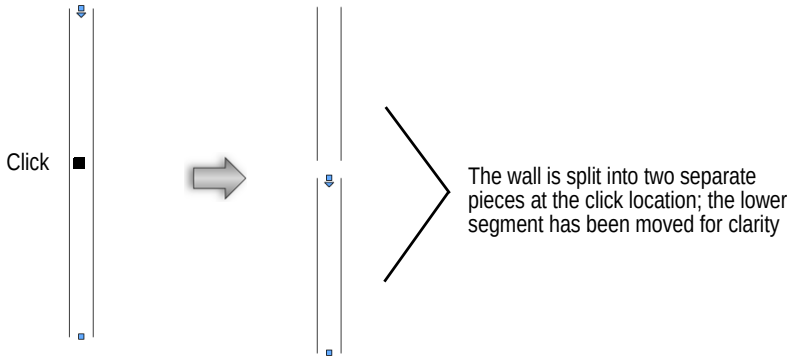
Splitting Objects by Point

To split an object at a specific point:

1. Click the **Split** tool from the Basic palette.
2. Click the **Point Split** mode from the Tool bar.
3. Click on the object at the point to be split.

   If multiple objects are within range of the specified point, the Select Split Candidate dialog box opens.

   Highlight the object to split by using the **Next** and **Prev** buttons.
4. The object is split into two pieces.



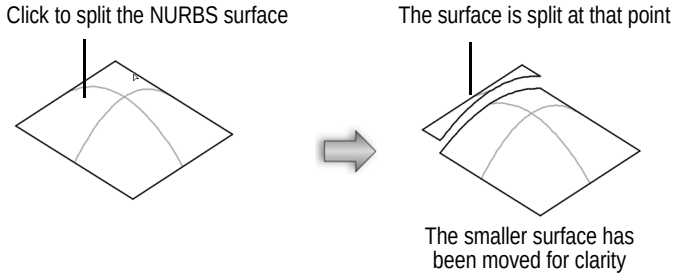
If the object is closed, as in a circle or rectangle, the object is converted so that its endpoints meet unjoined at the split. Some objects are not truly closed, and instead break into two segments.

Splitting NURBS Surfaces by Point

To split a NURBS surface in point split mode:

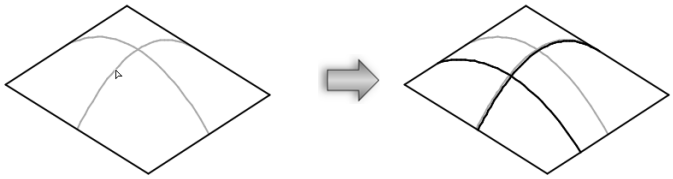
1. Click the **Split** tool from the Basic palette.
2. Click the **Point Split** mode from the Tool bar.
3. Click on the NURBS surface to split.

   See “Selecting the Edges and Faces of a Solid” on page 324 for information on selecting surfaces.



The surface is split by iso-parametric curves passing through the click point along U and V parametric directions.

If the split point is on an existing iso-parametric curve, the surface is split in both directions (U and V).



Clicking once on the iso-parametric curve (shown in light gray) with the **Split** tool creates four split surfaces (in black)

Line Split Mode
Line Trim Mode
Splitting or Trimming NURBS Surfaces

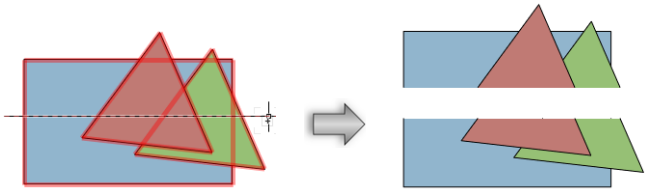
Line Split Mode

The **Line Split** mode splits 2D objects, NURBS curves/surfaces, solids, and viewports along a screen plane cutting line. After the split, all the objects remain on the drawing.

To split an object(s) along a cutting line:

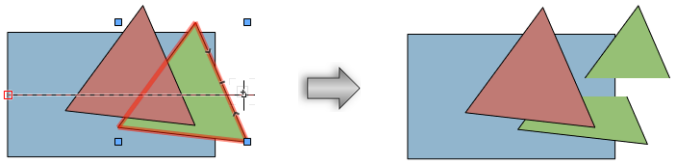
1. Click the **Split** tool from the Basic palette.
2. Click the **Line Split** mode from the Tool bar.
3. Click the **All Objects** or **Currently Selected Objects** mode from the Tool bar.
4. Draw a line through the object(s) to split; the object(s) to be split is highlighted.

   The object(s) is split by the line, and all parts remain in place on the drawing.



All Objects mode

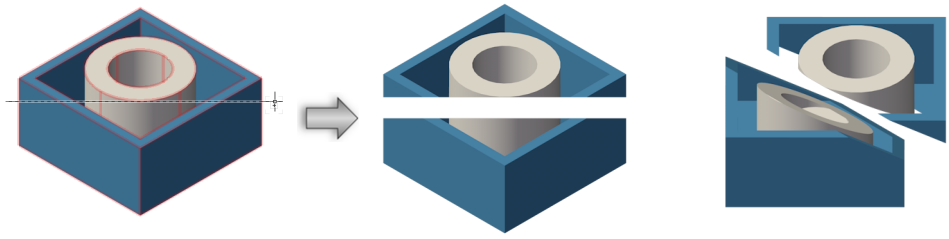
The split objects have been moved for clarity



Currently Selected Objects mode

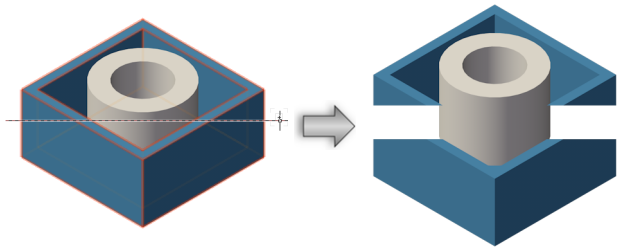
The split object has been moved for clarity

Solids can also be split in **Line Split** mode.



All Objects mode

Two solid sections have been moved and the image rotated for clarity



Currently Selected Objects mode

One solid section has been moved for clarity

Point Split Mode
Line Trim Mode
Splitting or Trimming NURBS Surfaces

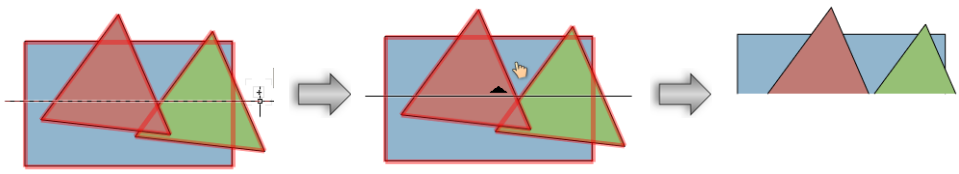
Line Trim Mode

The **Line Trim** mode splits 2D objects, NURBS curves/surfaces, solids, and viewports along a screen plane cutting line; it keeps a specified side, and trims away the other side.

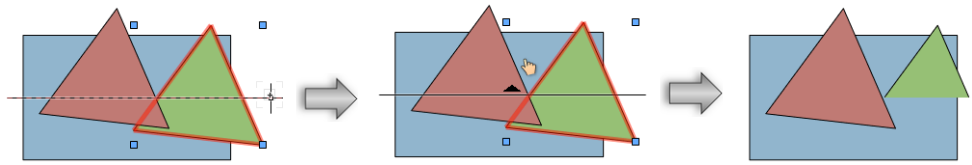
To split and trim an object along a cutting line:

1. Click the **Split** tool from the Basic palette.
2. Click the **Line Trim** mode from the Tool bar.
3. Click the **All Objects** or **Currently Selected Objects** mode from the Tool bar.
4. Draw a line through the object(s) to split; the object(s) to be split is highlighted.
5. An arrow points to the side to be kept. Click to indicate which side of the split line to keep; the other side is trimmed away.

   The object(s) is split by the line, and the indicated side remains.

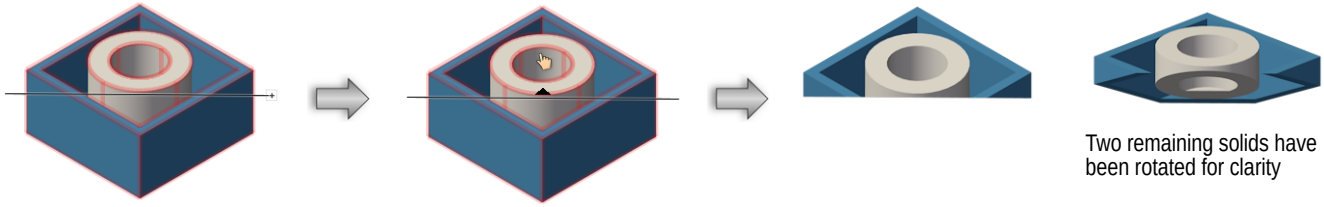


All Objects mode



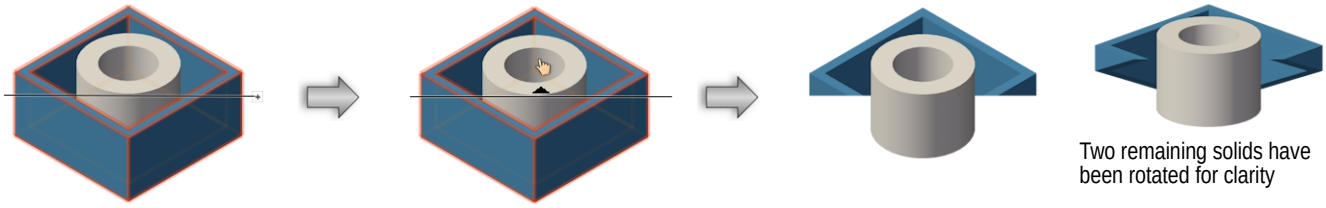
Currently Selected Objects mode

Solids can also be split in **Line Trim** mode.



All Objects mode

Two remaining solids have been rotated for clarity



Currently Selected Objects mode

Two remaining solids have been rotated for clarity

Point Split Mode
Line Split Mode
Splitting or Trimming NURBS Surfaces

Creating Fillets and Chamfers

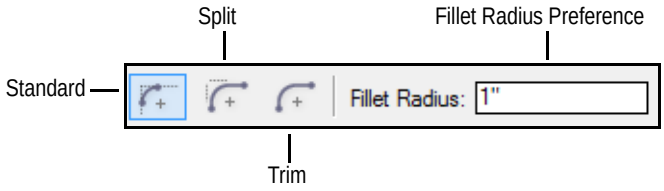
Fillet Tool

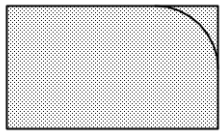
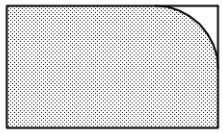
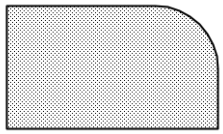
The **Fillet** tool adds a highly specific fillet (arc) between two objects in the drawing, making each of the fillet's end points tangent to one of the objects. Apply fillets to lines, rectangles, polygons, polylines, circles, circular arcs, NURBS curves, 3D polygons, and walls. With rectangles, polygons, or polylines, the tool places a fillet between adjacent sides of the object. Holes in polylines can be filleted. In addition, this tool trims or splits objects at the fillet's end points by selecting various fillet modes.

A fillet cannot be placed between parallel or concentric objects. If "split" or "trim" fillets are placed between a mixture of objects that can and cannot be split/trimmed, the fillet works only on the objects that can be split/trimmed, and ignores the others.

To place a fillet:

1. Click the **Fillet** tool from the Basic palette.
2. Click the desired mode from the Tool bar.



Mode	Description
Standard	Places a fillet without affecting the original object; the fillet and the object must be grouped to form a single object 
Split	Places a fillet and splits the filleted object. If the fillet is between two objects, the fillet and the objects must be grouped to make a single object. If filleting the corners of a polygon, the fillet takes the place of the corner and becomes a part of the object. 
Trim	Places a fillet and trims the filleted object. If the fillet is between two objects, the fillet and the objects must be grouped to make a single object. If filleting the corners of a polygon, the fillet takes the place of the corner and becomes a part of the object. 

3. Enter the **Fillet Radius** to use in the Tool bar.
4. Click the object where the fillet will begin.
5. Click the object where the fillet will end.

   To apply the fillet to all adjacent sides of an object, double-click.

   If an object is too short, it extends to match the selected fillet radius.

   Click here for a video tip on this topic (Internet access required).

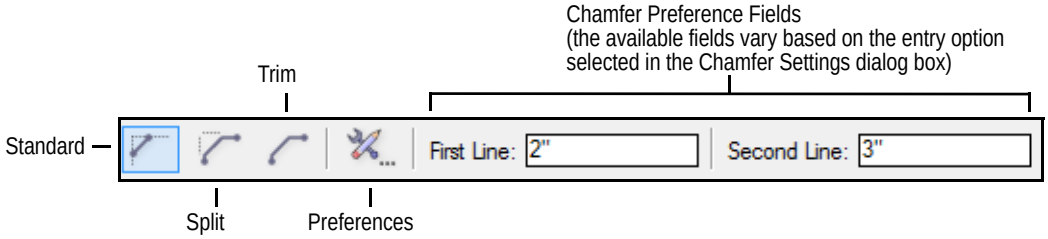
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Chamfer Tool
Filleting Solid Edges

Chamfer Tool

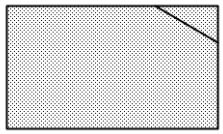
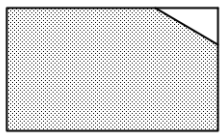
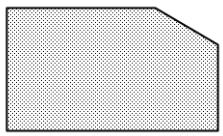
The **Chamfer** tool places a chamfer, or line, between two objects or adjacent sides of an object, including rectangles, NURBS curves, polygons, 3D polygons, polylines, or line segments. A chamfer cannot be placed between parallel lines or NURBS curves.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The tool has three modes: chamfers can be placed without affecting existing objects, they can split the existing objects at the chamfer endpoints, or they can trim (or extend) the existing lines to the chamfer endpoints.



To create a chamfer:

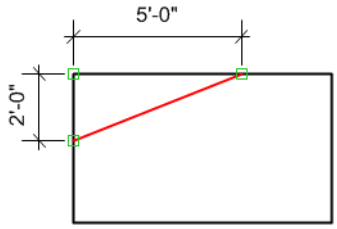
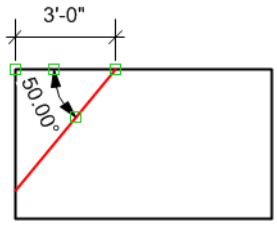
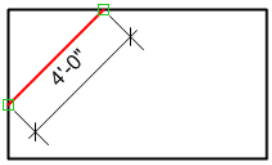
1. Click the **Chamfer** tool from the Basic palette.
2. Select the mode from the Tool bar.

Mode	Description
Standard	Places a chamfer without affecting the original object; to create a single object, group the chamfer and the chamfered object together 
Split	Places a chamfer and splits the chamfered objects. This mode extends lines, if needed, for the chamfer to connect. To create a single object, group the chamfer and the chamfered object together. If the corner of a polygon is chamfered, the chamfer takes the place of the corner. 
Trim	Places a chamfer and trims the chamfered objects. If the corner of a polygon is chamfered, the chamfer takes the place of the corner. This mode extends lines, if needed, for the chamfer to connect. 

3. To specify the chamfer size with a different method (for example, by a set chamfer length), click **Preferences** from the Tool bar. The Chamfer Settings dialog box opens. Enter the appropriate values and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Entry Options	Select a method for specifying the chamfer size; once selected, the required entry fields also display on the Tool bar for easy entry

Parameter	Description
First and Second Lines	Enter the distances from the end of the **First Line** and **Second Line** at which to place the ends of the chamfer line 
First Line and Angle	Enter the distance from the end of the **First Line** at which to place one end of the chamfer line, and enter the **Angle** between the First Line and the chamfer line 
Chamfer Line Length	Enter the **Chamfer Line Length** 

4. Fields on the Tool bar indicate where the chamfer will be placed, according to the current settings in the chamfer preferences. Change the default values if necessary.
5. Click the object where the chamfer will begin.
6. Click the object where the chamfer will end.

   The chamfer is drawn according to the mode selection.

   Click here for a video tip on this topic (Internet required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Fillet Tool
Chamfering Solid Edges

Editing Object Surfaces

Editing 2D Object Surfaces

There are four commands for editing 2D and 2D/3D hybrid object surfaces: Intersect Surface, Add Surface, Combine into Surface, and Clip Surface. The Intersect, Add, and Clip commands have an equivalent context menu command accessed by a Ctrl-click (Mac) or right-click (Windows). Use these commands with the following objects.

Vectorworks Fundamentals Objects

- 2D primitive objects that can be filled and are not grouped (arcs, polygons, polylines, ovals, circles, rectangles, rounded rectangles)
- Revision clouds
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Floors
- Roof faces
- Pillars

Vectorworks Design Series Objects

- Ceiling grids
- Massing models
- Property lines
- Site modifiers
- Spaces
- Parking Areas
- Slabs
- Plants
- Stipples
- Redlines
- Seating layouts
- Hardscapes
- Stages

These commands work in any view, as long as the objects involved are in the same plane. If a combination of 2D and 2D/3D hybrid objects (such as floors or pillars) will be used in an operation, the view must be set to Top/Plan. In Top/Plan view, screen plane and layer plane are considered to be co-planar.

The equivalent 3D commands operate on solids and can be used in a view other than Top/Plan.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Intersect Surface
Add Surface
Combine into Surface
Clip Surface
Editing 3D Object Surfaces

Intersect Surface

The **Intersect Surface** command provides an easy way to create a new object that is the exact size and shape of the overlapping area of two co-planar objects.

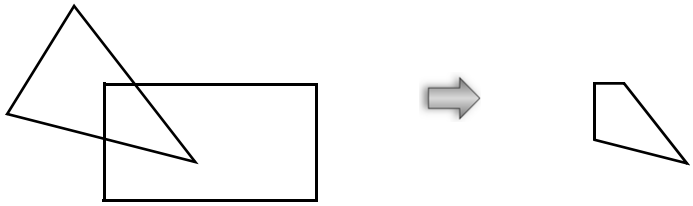
To intersect surfaces:

1. Select the two overlapping objects to use to create a third object.

   The new object's properties are based on the object on the bottom of the stack of objects. If the bottom object is a 2D primitive (such as a rectangle or circle), the new object will have its attributes. If the bottom object is something other than a 2D primitive (such as a floor or pillar), the new object will be the same type, with the same properties. If necessary, use the **Send** command to stack the objects to produce the desired attributes or object type.

2. Select **Modify > Intersect Surface**.

   The new object is placed directly on top of the two original intersecting objects. To see the new object, select it and drag it to the side.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing 2D Object Surfaces
Editing 3D Object Surfaces

Add Surface

The **Add Surface** command creates a single object from two or more co-planar objects, as long as all of the following are true:

- The objects are not symbols.
- The objects touch or overlap each other.
- The objects are not locked or grouped.

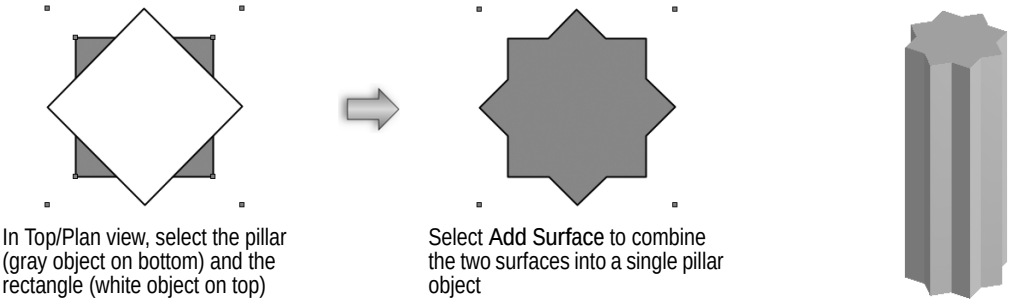
Note that any open polygons will be converted to closed polygons.

To add surfaces:

1. Select the two or more objects to be combined.

   The new object's properties are based on the object on the bottom of the stack of objects. If the bottom object is a 2D primitive (such as a rectangle or circle), the new object will have its attributes. If the bottom object is something other than a 2D primitive (such as a floor or pillar), the new object will be the same type, with the same properties. If necessary, use the **Send** command to stack the objects to produce the desired attributes or object type.

2. Select **Modify > Add Surface**.



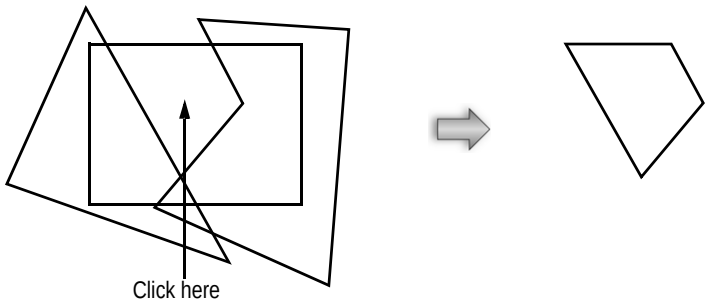
In Top/Plan view, select the pillar (gray object on bottom) and the rectangle (white object on top)

Select Add Surface to combine the two surfaces into a single pillar object

Editing 2D Object Surfaces
Editing 3D Object Surfaces

Combine into Surface

The **Combine into Surface** command creates a new object that is formed from a group of objects. The objects must currently intersect and form a closed polygon or polyline shape. Depending on the types of objects selected and the location of the mouse click, you can create several different polygons or polylines from the same selection of objects. For example, with this set of three objects, the following polygon can be created:



To combine surfaces:

1. Select the two or more closed objects to use to create a new polygon or polyline.
2. Select **Modify > Combine into Surface**.

   The cursor changes into a paint bucket.
3. Place the paint bucket inside the area to be combined and click.

   A single polygon or polyline object is created from the selected objects. The new object uses the current attributes.

Editing 2D Object Surfaces
Editing 3D Object Surfaces

Clip Surface

The **Clip Surface** command trims the bottom object in a selection so that any areas overlapped by the top object are cut out of it. Objects must be co-planar. Multiple objects can be used as clipping objects in one operation. Symbols and grouped objects cannot be clipped or be used as clipping objects.

If there is a stack of more than two overlapping objects, then each object under the clipping object (the top object in the stack) will be clipped.

There are two important things to remember about this command:

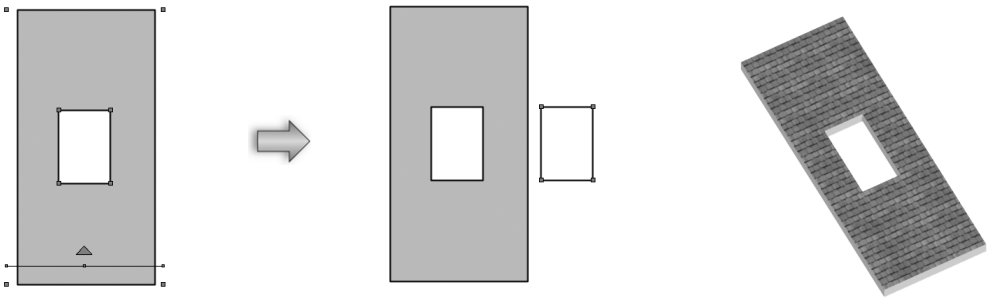
- If the object to be clipped is an open polygon, it is automatically converted to a closed polygon before it is clipped.
- Depending on the objects selected, the command may change the bottom object's type; for example, if a hole is clipped into a rectangle, the "clipped" rectangle is automatically changed into a polyline.

To clip a surface:

1. Ensure that the object to be clipped is the bottom object.

   If necessary, change the objects' order with the **Send** command (see "Changing Object Stacking Order" on page 1010).
2. Select the object to be clipped.
3. Select the clipping object.
4. Select **Modify > Clip Surface**.

   The bottom object is clipped by the clipping object; the clipping object can be deleted if it is no longer needed.



In Top/Plan view, select the roof face (gray object on bottom) and the rectangle (white object on top)

Select Clip Surface, and then move the rectangle clipping object to reveal the modified roof face

Editing 2D Object Surfaces
Editing 3D Object Surfaces

Editing 3D Object Surfaces

The 3D editing commands are similar to the 2D **Add** or **Clip Surface** commands, but for 3D objects. For 3D objects, the editing commands are: Intersect Solids, Add Solids, Subtract Solids, and Section Solids. Each of these commands has an equivalent context menu command accessed by a Ctrl-click (Mac) or right-click (Windows).

Intersect Solids creates a single model from the volume created where two or more 3D objects intersect. **Add Solids** joins two or more 3D objects into a single model. **Subtract Solids** cuts (subtracts) a 3D object(s) from another 3D object, creating a new model. **Section Solids** discards a portion of solids or NURBS surfaces, allowing the creation of planar or stepped sections through a solid or surface.

These commands work with the following solid objects: extrudes, multiple extrudes, straight walls, sweeps, meshes, solid primitives (cylinders, hemispheres, spheres, and cones), and objects created using the **3D Polygon**, **Extruded Polygon**, and **Extruded Rectangle** tool, provided the following applicable conditions are met.

Object	Criteria
Sweeps	Cannot contain lines, or be swept around a locus which is between the left and right bounds of the swept 2D primitive; if not swept around a locus, should have a vertical segment on the left edge
Helical Sweep	Must be swept around a locus outside the left and right bounds of the object
Multiple Extrude	Must have planar polygons
Meshes	Cannot have interpenetrating polygons; every edge of every polygon in the mesh must be shared with one other polygon
Walls	Must not have symbols that extend above the top or below the bottom of the wall

2D objects must have a fill applied prior to being converted to 3D to be considered a solid object. If the converted 3D object did not have a fill applied prior to conversion, it can be added using the **Enter Group** command to return to the original 2D object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Add Solids
Intersect Solids
Subtract Solids
Section Solids
Editing 2D Object Surfaces

Add Solids

The **Add Solids** command joins two or more 3D objects into a single model.

To add solids in 3D:

1. Select two or more 3D objects to combine.
2. Select **Model > Add Solids**.

   A single solid model is created from the objects.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing 3D Object Surfaces
Editing 2D Object Surfaces

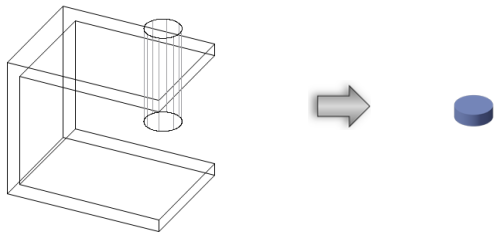
Intersect Solids

The **Intersect Solids** command creates a single model from the volume created where two or more 3D objects intersect.

To intersect solids:

1. Select the two or more 3D objects to combine.
2. Select **Model > Intersect Solids**.

   A single solid model is created from the objects; it is the size and shape of the overlapping volume of the selected objects.



Editing 3D Object Surfaces
Editing 2D Object Surfaces

Subtract Solids

The **Subtract Solids** command cuts (subtracts) one or more 3D objects from another 3D object, creating a new model.

The subtracting object should extend beyond the surface of the original object.

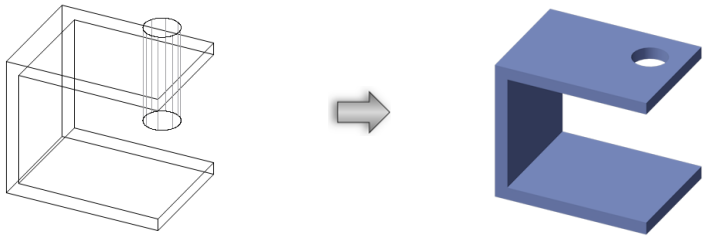
To subtract solids:

1. Select both the object(s) to subtract and the object to subtract from (the base object).
2. Select **Model > Subtract Solids**.

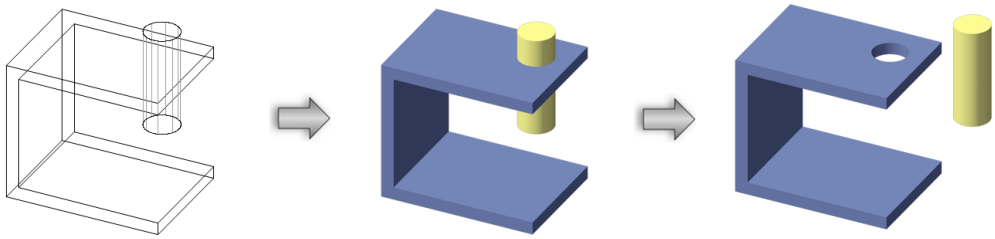
   The Select Object dialog box opens.
3. Use the forward and back arrows to select the base object, which is shown with a thick outline.

   Deselect **Retain Subtracting Objects** to automatically delete the subtracting object, or select **Retain Subtracting Objects** to keep the object in place.
4. Click **OK**.

   If **Retain Subtracting Objects** is not selected, a single solid model is created, with the subtracting object deleted.



If **Retain Subtracting Objects** is selected, a mass equal to the subtracting object is removed from the base object, but the subtracting object remains in place and can be manipulated independently.



Editing 3D Object Surfaces
Editing 2D Object Surfaces

Section Solids

The **Section Solids** command discards a portion of solids or NURBS surfaces, allowing the creation of planar or stepped sections through a solid or surface. The sectioned surface can be "marked" by the sectioning surface color.

The sectioning surface must be larger than the base object.

To section a solid:

1. Select both the object to be sectioned (the base object), and the sectioning surface.

   The operation will delete the portion of the base object on the same side as the sectioning surface's normal; select **Show Normal** in the Object Info palette to display the surface normals (see "Displaying Surface Normals" on page 326). Click **Reverse Normal** in the Object Info palette if needed to achieve the desired results.

2. Select **Model > Section Solids**.

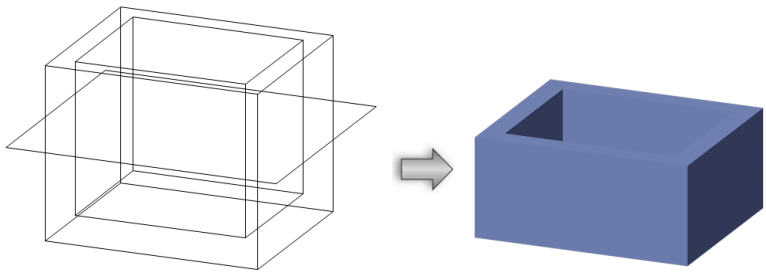
   The Select Object dialog box opens.

3. Select the sectioning surface. By default, the most recently created object is highlighted, but a different object can be selected by clicking the arrows.

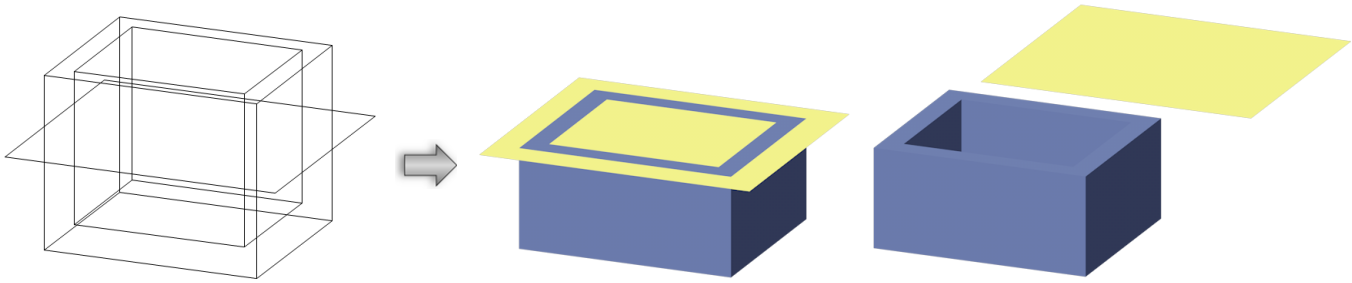
   Select **Retain Sectioning Object** to keep the object in place, or deselect **Retain Sectioning Object** to automatically delete the sectioning object.

4. Click **OK**.

   If **Retain Sectioning Object** is not selected, a solid section is created, with the sectioning surface deleted.



   If **Retain Sectioning Object** is selected, a solid section is created, but the sectioning surface remains in place and can be manipulated independently.



The solid section parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Width/Depth/Height (display only)	Displays the parameters of the solid section surface
Reverse Section Side	Switches the remaining side of the solid being sectioned
Use Section Color	Applies the color of the sectioning surface to the sectioned surface

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing 3D Object Surfaces
Editing 2D Object Surfaces

Drafting Aids

The Drafting Aids menu contains several commands that create new arcs, lines, or rectangles that are based on existing objects, or simplify existing objects, to make drafting faster and easier.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Arc into Segments
Line into Segments
Create Dividing Lines
Even Divide
Simplifying Polygons and Polylines

Arc into Segments

The **Arc into Segments** command converts a selected arc or circle into an equal number of segments, or divides the arc or circle into segments of a given length. The segments can be drawn as lines or polygons; the original object remains unchanged.

To convert a circle or arc into segments:

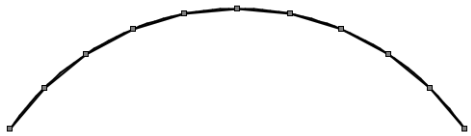
1. Select the arc or circle to be converted.
2. Select **Modify > Drafting Aids > Arc into Segments**. The Arc into Segments dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Method	
Number of Segments	Draws the specified number of equal segments along the arc
Segment Length	Draws segments of the specified length along the arc

Parameter	Description
Options	
Draw Polygon	Draws the segments as polygons
Draw Lines	Draws the segments as lines

3. Click **OK** to create the segments.



Line into Segments
Create Dividing Lines
Even Divide
Simplifying Polygons and Polylines

Line into Segments

The **Line into Segments** command converts a selected line into the indicated number of equal length segments. The original line can be converted, or an identical line placed on top of the original and converted.

To convert a line into segments:

1. Select the line to be converted.
2. Select **Modify > Drafting Aids > Line into Segments**.

   The Line into Segments dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Place loci	Places loci at the segment divisions
Break Line Into Segments	Creates segments from the original line, converting it; deselect to place loci only
Leave Original Line Intact	Retains the original line, and creates segments from a copy of the line
Number of Segments	Specifies the number of segments to create

3. Set the parameters and click **OK** to draw either a new segmented line or convert the selected one.

Arc into Segments
Create Dividing Lines
Even Divide
Simplifying Polygons and Polylines

Create Dividing Lines

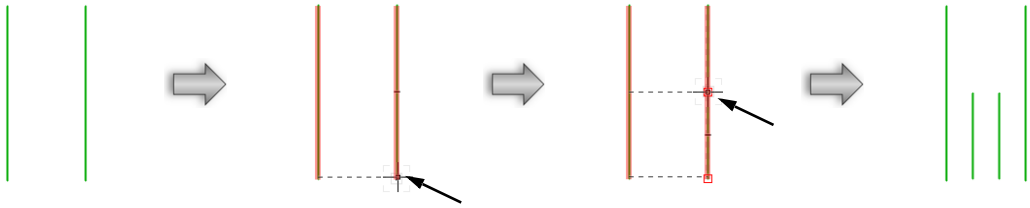
The **Create Dividing Lines** command creates lines that evenly divide the space between two existing lines. The existing lines can be angled or parallel to each other.

To create dividing lines:

1. Select **Modify > Drafting Aids > Create Dividing Lines**.

2. Select the two lines to create lines between.
3. Click to define first the starting point and then the ending point of the dividing lines. A perpendicular line (for parallel lines) or circle (for angled lines) indicates the edge along which the lines will start and end. If the starting point and ending point are on the same line or circle, loci will be created instead of lines.
4. When the Dividing Lines Settings dialog box opens, enter the number of equal divisions to create between the existing lines.
5. Click **OK** to create evenly spaced lines (or loci) between the existing lines.

Creating divisions between parallel lines

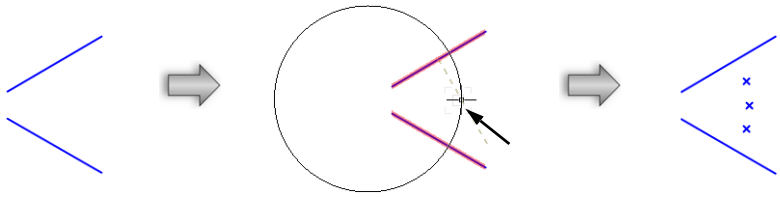


After you select the two lines, the next click indicates the starting edge for the dividing lines

Click another point to indicate the ending edge for the dividing lines

The specified number of divisions are created; in this case, two lines are added to create three divisions

Creating divisions between angled lines



After you select the two lines, click two points anywhere on the same indicator circle to create loci instead of lines

The specified number of divisions are created; in this case, three loci are added to create four divisions

Arc into Segments
Line into Segments
Even Divide
Simplifying Polygons and Polylines

Even Divide

The **Even Divide** command divides lines, arcs, circles, and rectangles into the specified number of subdivisions. You can choose whether or not to retain the original object.

To evenly divide lines, arcs, circles, and rectangles:

1. Select one or more objects to divide.
2. Select **Modify > Drafting Aids > Even Divide**.

   The Even Divide dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Number of Divisions for Line, Circle and Arc	
Number of Divisions	Specifies the number of even subdivisions to create from each selected line, circle, or arc (must be greater than 1)
Number of Divisions for Rectangle	
Number of Divisions in Width	Specifies the number of even subdivisions to create along the width of each selected rectangle; enter 1 to create rectangles of the same width as the original rectangle
Number of Divisions in Height	Specifies the number of even subdivisions to create along the height of each selected rectangle; enter 1 to create rectangles of the same height as the original rectangle
Original Object	
Retain	Select to retain the original object from which the subdivisions are created; deselect to delete the original object
Leave Selected	If the original object is retained, select this option to leave both the original object and the newly-created objects selected after the division operation. Deselect this option to leave only the newly-created objects selected after the operation.

3. Set the parameters and click **OK** to create new objects that are even subdivisions of the selected object(s).

In this example, four rectangles were created from the original rectangle; the width was divided into four even sections, and the height remained in one section (one rectangle moved for clarity)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Arc into Segments
Line into Segments
Create Dividing Lines
Simplifying Polygons and Polylines

Simplifying Polygons and Polylines

The **Simplify Polys** command simplifies 2D polygons, 3D polygons, and polylines by reducing the number of corner vertices. This operation can reduce the file size and shorten processing time while having a minimal effect on the shape of the polygons/polylines themselves. There are two methods for simplifying polygons/polylines: by the maximum deviation or by the minimum distance.

To simplify a polygon(s) or polyline(s):

1. Select the polygon(s) and/or polyline(s) to simplify.
2. Select **Modify > Drafting Aids > Simplify Polys.**

   The Simplify Polys dialog box opens.

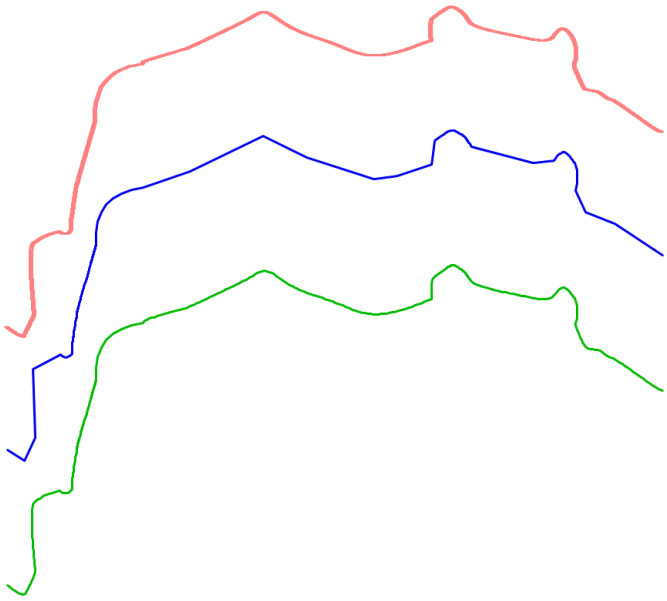
Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Simplify by	
Maximum Deviation	Select to remove the corner vertices within a specified maximum tolerable deviation between the original and the simplified polygon(s)/polyline(s)
Deviation	Specify the maximum deviation value. A deviation value of zero removes all collinear vertices.
Minimum Distance	Select to remove corner vertices located equal to or less than a specified minimum distance from adjacent vertices
Distance	Specify the minimum distance value. A distance of zero removes all extra vertices existing at the same location.

3. Set the parameters and click **OK** to simplify the polygon(s)/polyline(s).

   The **Simplify Polys** command does not remove start and end vertices, and does not remove non-corner vertices unless multiple vertices exist at the same location. If you select the **No Smoothing** command first, **Simplify Polys** will then work on an object with these vertices.

   The command works for stand-alone polygons/polylines. To simplify polygons/polylines located inside objects such as groups and symbol definitions, you must enter the object and simplify the polygon or polyline while in object editing mode (see “Object Editing Mode” on page 1004).



Drafting Aids
Arc into Segments
Line into Segments
Create Dividing Lines
Even Divide
Simplifying 3D Polygons
Smoothing Objects

Applying Object Attributes

Attributes are characteristics that can be applied to 2D planar and screen objects in a drawing, including fill style, pen style, opacity, line thickness, and line end markers. Fill styles include solid colors, patterns (including foreground and background colors), hatches, tiles, gradients, and images. Pen styles include solid colors, patterns (including foreground and background colors), and line types.

There are various ways to apply attributes to objects:

- Use the Attributes palette to apply attributes to a selected object.
- Use the Attributes palette to set default attributes that will be applied as objects are created.
- Use the **Eyedropper** tool to transfer attributes from one object to another.
- Set up a class to use certain attributes when objects are created in that class or when the class is assigned to an existing object.

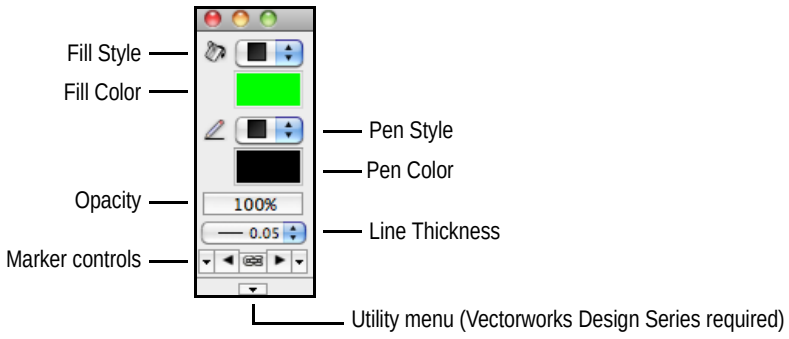
The attributes available on the Attributes palette can be customized. Fill and line patterns, line thickness, and line end marker style attributes can all be adjusted as needed; to reuse the custom attributes in other files, save them to a template file. Attributes that are resources (hatches, tiles, gradients, images, and line types) are even more flexible; they can easily be created, edited, reused in other files, and shared with coworkers. In addition, hatches, tiles, gradients, and images can be mapped to suit a specific object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Attributes Palette

The Attributes palette applies attributes to objects, and displays the attributes currently applied to a selected object.

Select **Window > Palettes > Attributes** to open the Attributes palette.



Attributes can be applied to most 2D planar and screen objects, with a few exceptions. Line end markers can only be applied to open objects, such as lines, arcs, and polylines. Only pen color and opacity can be applied to text; however, a fill can be applied to the text box (the background behind the text).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

If you are using the same attribute settings for several objects, create a class for those objects and apply the class attributes at creation. Class attributes can also be set for selected objects; see “Setting Class Attributes” on page 181.

Most of the attribute types (fill, pen, opacity, line thickness, and marker) are specified directly from the Attributes palette. Some of the settings available on the palette can be customized; fill and line patterns, line thickness, and line end marker styles can all be adjusted as needed.

Line types, and hatch, tile, gradient, and image fills are resources; you can select them from the default content in the Attributes palette (see “Resource Libraries” on page 219), or you can create or import custom resources and apply them from the Resource Browser.

In Vectorworks Design Series products, the Attributes palette has a utility menu that sets default and global attributes.

Setting Default Attributes

The default attributes for a document are the attributes that are automatically applied to each new object when it is created, unless a class attribute is assigned. These are the settings that display on the Attributes palette when no objects are selected.

When you first open a new document, set the default attributes to those you use most often. For example, you might adjust the fill and line colors, and line thickness.

The default attributes apply to the current document only. In addition, custom patterns that you create cannot be used in other documents. To save a set of default attributes or custom patterns for future use, set the attributes as desired and save the document as a template.

To set the default attributes for a document:

1. Ensure that no objects are selected.
2. Select **Window > Palettes > Attributes**.

   The Attributes palette opens.
3. Select the desired attributes. Objects created after that point use those attributes by default.

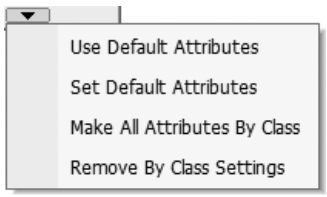
   The defaults can also be set with the **Eyedropper** tool (see “Transferring Attributes” on page 1095).

D Setting Global and Default Attributes

The Attributes palette includes a utility menu for making changes to all the attributes at one time. A set of default attributes can be saved and then restored, and attributes can quickly be set to be “By Class,” or have all “By Class” settings removed.

To access the Attributes palette utility menu:

1. From the Attributes palette, click the **Utility Menu** button to open the **Utility** menu.



Command	Description
Use Default Attributes	Restores the default Attributes palette settings
Set Default Attributes	Saves all current parameter settings in the Attributes palette as the default set

Command	Description
Make All Attributes By Class	Sets all Attributes palette parameters to be determined by the object's class (see "Setting Class Attributes" on page 181)
Remove By Class Settings	Removes any "by class" settings for all Attributes palette parameters

2. Select a utility command to set the default attributes or change the Attribute palette parameter settings.

Applying Attributes to Existing Objects

To apply attributes to existing objects:

1. Select the object or objects.
2. Select **Window > Palettes > Attributes**.

   The Attributes palette opens.
3. Set the desired attributes for the object(s) from the Attributes palette.

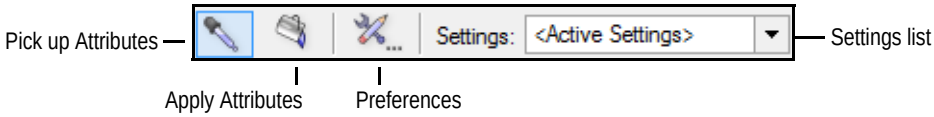
   The object's attributes are updated.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Templates
Transferring Attributes
Fill Attributes
Pen Attributes
Opacity Attributes
Marker Attributes
Using Line Types
Applying Colors

Transferring Attributes

The **Eyedropper** tool transfers attributes from one object to another in a single step, including fill, pen, line, text, wall, and other attributes.



Mode	Description
Pick up Attributes	Selects an object's attributes
Apply Attributes	Transfers selected attributes to another object
Preferences	Sets the default parameters to be used for the tool
Settings	Sets the group of parameters to be used for the tool-either the currently active settings, or a selection from the list of saved settings

Press the Option (Mac) or Ctrl (Windows) key to switch between the Pick Up and Apply modes.

 To transfer object attributes:

1. Click the **Eyedropper** tool from the Basic palette.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Click **Preferences** from the Tool bar to specify the set of attributes to transfer. The Eyedropper Preferences dialog box opens.

   Alternatively, select a set of saved attributes from the Settings list on the Tool bar.

3. Specify the attributes to be selected and applied by the **Eyedropper** tool, and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Saved Settings Options	
Settings	Select <Active Settings> to use the attributes currently selected in the dialog box; to use a set of saved attributes, select them from the pull-down list
Save	Opens a dialog box to name and save the currently selected attributes so that they can quickly be selected as a set
Manage	Opens the Saved Settings dialog box to rename or delete sets of saved tool attributes
All	• **Checked box:** All attributes in the group are selected; click to deselect all attributes in the group. • **Empty box:** No attributes in the group are selected; click to select all attributes in the group for transfer. • **Box displays (Windows) or (Mac):** One or more attributes in the group are not selected; click to select all attributes in the group for transfer.
Fill Attributes	Transfers fill attributes, including foreground and background colors, style (pattern, hatch, tile, gradient, or image fill), and texture
Pen Attributes	Transfers pen foreground and background colors and style (solid, pattern, or line type)
Line Attributes	Transfers line weight (thickness), presence of markers, and marker attributes
Text Attributes	Transfers text attributes to another text object, dimension, or plug-in object, including font, size, font style, alignment, spacing, tab stops, and tracking. Only font, size, and font style settings can be transferred to dimension and plug-in objects.
Wall Attributes	Transfers wall thickness and component settings for walls and round walls; components define wall thickness, so components cannot be transferred without also transferring the thickness parameter
Viewport Attributes	Transfers viewport attributes to another viewport, including class and layer visibility settings, class and layer attribute overrides, and rendering properties (background and foreground rendering modes and options, lighting options, Renderworks background, light overrides and Renderworks styles). **Other Properties** refers to the remaining viewport options such as view, projection, scale, and advanced properties. Viewport attributes can be transferred between files; class, layer, and other resources specified in an attribute override are resolved by the name of the class, layer, or resource, respectively.
Other Attributes	
Record	Transfers the database record attributes
Plug-in Parameters	Transfers plug-in parameters between plug-in objects, including objects inserted in walls
Class	Selects the class for transfer (makes the target object the same class as the source object)

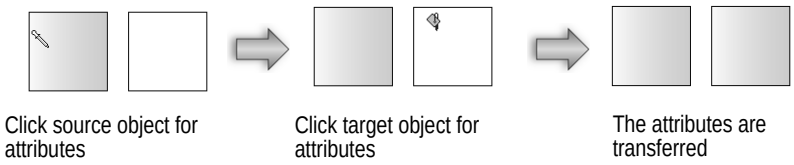
Parameter	Description
IFC Data (Vectorworks Architect or Landmark required)	Transfers IFC data to another object, while preserving unique IFC identifiers. If the object receiving the data does not yet have a unique identifier, it is automatically created.
Pick Up Sets Defaults	Changes the default Attributes palette settings to match those of the source object; objects created from then on use the attributes of the source object
Use Class Attributes	Transfers the "by-class" settings of the source object, provided the target object is in the same class as the source object
Object Opacity	Transfers an object's opacity setting

4. Click **Pick Up Attributes** from the Tool bar.

   The cursor changes to an eyedropper.
5. Click the object that is the source of the attributes.
6. Click **Apply Attributes** from the Tool bar.

   The cursor changes from an eyedropper to a paint bucket.
7. Click the target object for the attributes.

   The attributes are transferred to the object.



To transfer attributes between files, the target objects, classes, records, textures, and other attributes must already be present in the target file. The **Eyedropper** tool does not create objects or attributes.

Click here for a video tip on this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Attributes Palette
Managing Saved Settings

Managing Saved Settings

If you have saved attributes to use with the **Eyedropper** tool, use the **Manage** button on the Eyedropper Preferences dialog box to rename or delete these saved settings when needed.

To manage saved settings for the **Eyedropper** tool:

1. Select the **Eyedropper** tool from the Basic tool palette, and then click **Preferences** on the Tool bar. The Eyedropper Preferences dialog box opens.
2. Click **Manage**.

   The Saved Settings dialog box opens. Select the set of saved attributes to change.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Rename	Select and enter a new name for the set of attributes. If the entered name is already assigned to another set of attributes, you are prompted to confirm that you want to replace the existing set with the set being saved.
Delete	Select to delete the set of attributes. You are prompted to confirm that you want to delete the saved settings.

3. Click **OK** to close the Saved Settings dialog box. Click **OK** again to close the Eyedropper Preferences dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Transferring Attributes

Fill Attributes

Objects can be filled with a solid color, pattern (with foreground and background colors), hatch, tile, gradient, or image. Alternatively, set the object fill to None to create transparent objects with no fill.

Hatch, tile, gradient, and image fills can be customized to suit the object to which they are applied. The Attributes palette displays a button for each of these fills to open a settings dialog box for customizing the fill. In addition, you can use the **Attribute Mapping** tool to edit the size, position, and angle of an associative hatch, tile, gradient, or image fill that has been applied to an object (see "Mapping Fills with the Attribute Mapping Tool" on page 1123).

Fill Style	Description
None	No fill is applied
Solid	Applies a solid fill to the selected object(s); click the color box to select the fill color. To set the color by class, select the **Color by Class** option from the color palette.
Pattern	Applies a pattern fill to the selected object(s); click the pattern box to select the desired pattern, and then select the foreground color and background color from the color boxes next to the pattern. To set the pattern colors by class, select the **Color by Class** option from the color palette. To create custom patterns for this file, see "Creating Custom Patterns for Fill and Pen Styles" on page 1099.
Hatch	Applies a hatch fill to the selected object(s). Select the desired hatch from either the default content or the current file's content (see "Resource Libraries" on page 219). If there are no hatches defined in the document and default content is not enabled, you are prompted to add a default hatch definition. To customize the hatch fill for the selected object(s), click the **Fill Hatch Settings** button). If the fill of the selected object has been edited, the fill name is highlighted in blue and "(local map)" is appended to the name. For more information about how to create, edit, and customize hatches, see "Using Hatch Fills" on page 1104.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Fill Style	Description
Tile	Applies a tile fill to the selected object(s). Select the desired tile fill from either the default content or the current file's content (see "Resource Libraries" on page 219). If there are no tiles defined in the document and default content is not enabled, you are prompted to add a default tile definition. To customize the tile fill for the selected object(s), click the **Fill Tile Settings** button. If the fill of the selected object has been edited, the fill name is highlighted in blue and "(local map)" is appended to the name. For more information about how to create, edit, and customize tiles, see "Using Tile Fills" on page 1111.
Gradient	Applies a gradient fill to the selected object(s). Select the desired gradient from either the default content or the current file's content (see "Resource Libraries" on page 219). If there are no gradients defined in the document and default content is not enabled, you are prompted to add a default gradient definition. To customize the gradient fill for the selected object(s), click the **Fill Gradient Settings** button. For more information about how to create, edit, and customize gradients, see "Using Gradient Fills" on page 1115.
Image	Applies an image fill to the selected object(s). Select the desired image from either the default content or the current file's content (see "Resource Libraries" on page 219). If there are no images defined in the document and default content is not enabled, you are prompted to import an image definition. To customize the image fill for the selected object(s), click the **Fill Image Settings** button. For more information about how to create, edit, and customize images, see "Using Image Fills" on page 1119.
Class Style	The selected object(s) takes on the fill attribute set by the object's class. • If **Use at Creation** is enabled for the object's class, the object's fill style is automatically set by the class. • If **Use at Creation** is disabled for the class, the object only takes on the class style when the **Class Style** option is selected. See "Setting Class Attributes" on page 181.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Attributes Palette
Creating Custom Patterns for Fill and Pen Styles

Creating Custom Patterns for Fill and Pen Styles

There are 72 patterns available to use as a fill or pen style in the Attributes palette. You can customize patterns 36 through 71; patterns 0 through 35 cannot be edited. Customizations are saved with the file.

To reuse the custom patterns in other files, save the file as a template file. See "Creating Templates" on page 75 for more information.

To customize the patterns available in the current file:

1. Select **File > Document Settings > Patterns**.

   The Edit Patterns dialog box opens. Patterns 36 through 71 can be selected from the list; select a pattern and edit it as desired.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Click to show/hide the parameters.

Parameter	Description
Pattern	Select a pattern to edit from the **Pattern** list, and then click in the editing box on the left to add and delete pixels; the edits display in the preview window to the right of the editing box
Revert	Restores the pattern to its original settings (you must click this before you select another pattern or click **OK**)

2. When all edits are complete, click **OK** to save the customized patterns. The edited patterns are available from the Fill Pattern and Pen Pattern boxes on the Attributes dialog box, for the current file only.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Fill Attributes
Pen Attributes
Creating Templates

Pen Attributes

Pen attributes apply to object outlines, and can be a solid or pattern line of any thickness and color. Alternatively, apply a standard or custom line type, to add graphic elements to a line. Line types are resources that can include dashes, text, or any 2D object.

To apply color to text objects, select the Solid pen style, and set the appropriate color.

For more information about using color boxes in the Attributes palette, see “Applying Colors” on page 1132.

Pen Style	Description
None	No pen is applied
Solid	Applies a solid pen style to the selected object(s); click the color box to select the pen color. To set the color by class, select the **Color by Class** option from the color palette. Select the line thickness for the pen from the Line Thickness list. To create custom line thicknesses for this file, see “Creating Custom Thicknesses for Lines” on page 1102.
Pattern	Applies a patterned pen style to the selected object(s); click the fill pattern box to select the desired pattern, and then select the foreground color and background color from the color boxes next to the pattern. To set the pattern colors by class, select the **Color by Class** option from the color palette. To create custom patterns for this file, see “Creating Custom Patterns for Fill and Pen Styles” on page 1099.
Line Type	Applies a line type resource to the selected object(s). Click the color box to select the pen color. To set the color by class, select the **Color by Class** option from the color palette. Select the desired thickness from the Line Thickness list. To create line type resources for this file, see “Using Line Types” on page 1128.
Class Style	The object(s) takes on the pen attribute set by the object’s class. • If **Use at Creation** is enabled for the object’s class, the object’s pen style is automatically set by the class. • If **Use at Creation** is disabled for the class, the object only takes on the class style when the **Class Style** option is selected. See “Setting Class Attributes” on page 181.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Attributes Palette
Creating Custom Patterns for Fill and Pen Styles
Creating Custom Thicknesses for Lines
Using Line Types
Applying Colors

Opacity Attributes

In addition to the overall layer opacity settings described in “Setting the Design Layer Opacity” on page 168, an opacity setting is available for individual objects. Opacity can be applied to any type of 2D object, including planar objects, 2D walls, text, worksheet backgrounds, and plug-in objects that include a 2D object.

> If layer opacity is set in addition to object opacity, the results are additive; for example, a layer opacity of 50% and an object opacity of 50% will display the object at 25% opacity. Similarly, the opacity settings of nested and container objects are combined.

This feature is available on all Mac systems and on Windows systems that have the GDI+ imaging Vectorworks preference enabled. See “Vectorworks Display Preferences” on page 50.

To set the opacity of a 2D object:

1. Select the object(s) and click the **Opacity** button on the Attributes palette to open the Set Opacity dialog box.
2. Drag the **Opacity** slider to the left to decrease the opacity, or enter an opacity percentage (0-100) to the right of the slider. Alternatively, select **Use Class Opacity** to use the opacity value set by the object’s class.

   If **Use at Creation** is enabled for the object’s class, the object’s opacity is automatically set by the class. If **Use at Creation** is disabled for the class, the object only takes on the class style when the **Use Class Opacity** option is selected. Class opacity can be overridden in viewports.
3. If desired, click **Preview** to view the results of the opacity setting.
4. Click **OK** to apply the opacity setting.

The opacity setting applies to 2D objects only. For a similar effect in 3D, a transparency shader can be applied with the Renderworks product (see “Textures and Shaders” on page 1505).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Class Properties
Drawing Straight Walls
The Attributes Palette

Line Thickness Attributes

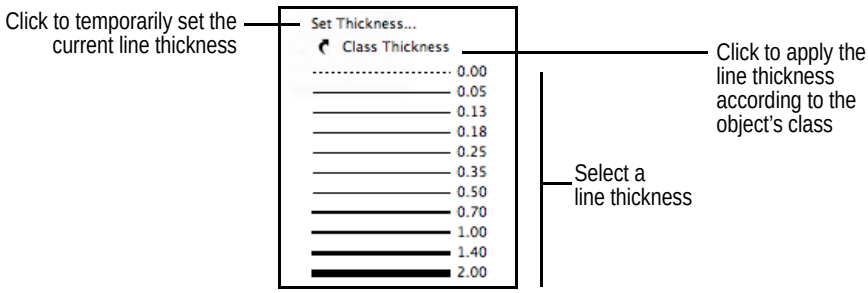
Line thickness attributes apply to the pen style of an object. Apply line thickness attributes to 2D and 3D objects.

From the Attributes palette, select the line thickness from the Line Thickness list.

To add, remove, or edit the available line thickness values, select **Tools > Options > Line Thickness**. See “Creating Custom Thicknesses for Lines” on page 1102 for more information. To temporarily adjust the current line thickness setting, click **Set Thickness** from the Line Thickness list. The Set Thickness dialog box opens; select the **Thickness Units**, enter the **Thickness Value**, and click **OK**.

Select **Class Thickness** to use the line thickness attributes set by the object’s class (see “Setting Class Attributes” on page 181).

- If **Use at Creation** is enabled for the object’s class, the object’s line thickness is automatically set by the class.
- If **Use at Creation** is disabled for the class, the object only takes on the class thickness when the **Class Thickness** option is selected.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Custom Thicknesses for Lines
Marker Attributes
The Attributes Palette

Creating Custom Thicknesses for Lines

You can customize the set of line thicknesses available from the Line Thickness list on the Attributes palette. The edited thicknesses are saved with your user preferences, so they are available from one Vectorworks session to the next.

To create custom line thicknesses for the Attributes palette:

1. Select **Tools > Options > Line Thickness**.

   The Preferred Line Thickness dialog box opens. Select the preferred line thickness criteria.

Click to show/hide the parameters.

Parameter	Description
Units	Select the **Units** to be used for the thickness of the edited line
New	Enter the new thickness value to replace the current thickness value listed to its left

2. When all edits are complete, click **OK** to save the customized set of line thicknesses.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Line Types
Marker Attributes
Pen Attributes
The Attributes Palette

Marker Attributes

You can apply line start and line end markers to open objects, including lines, arcs, polylines, 2D polygons, and freehand lines. You can also apply line end markers to the leader lines on callout and dimension objects. Dimensions have other default markers assigned as part of the dimension standard setup, but these cannot be applied or removed through the Attributes palette.

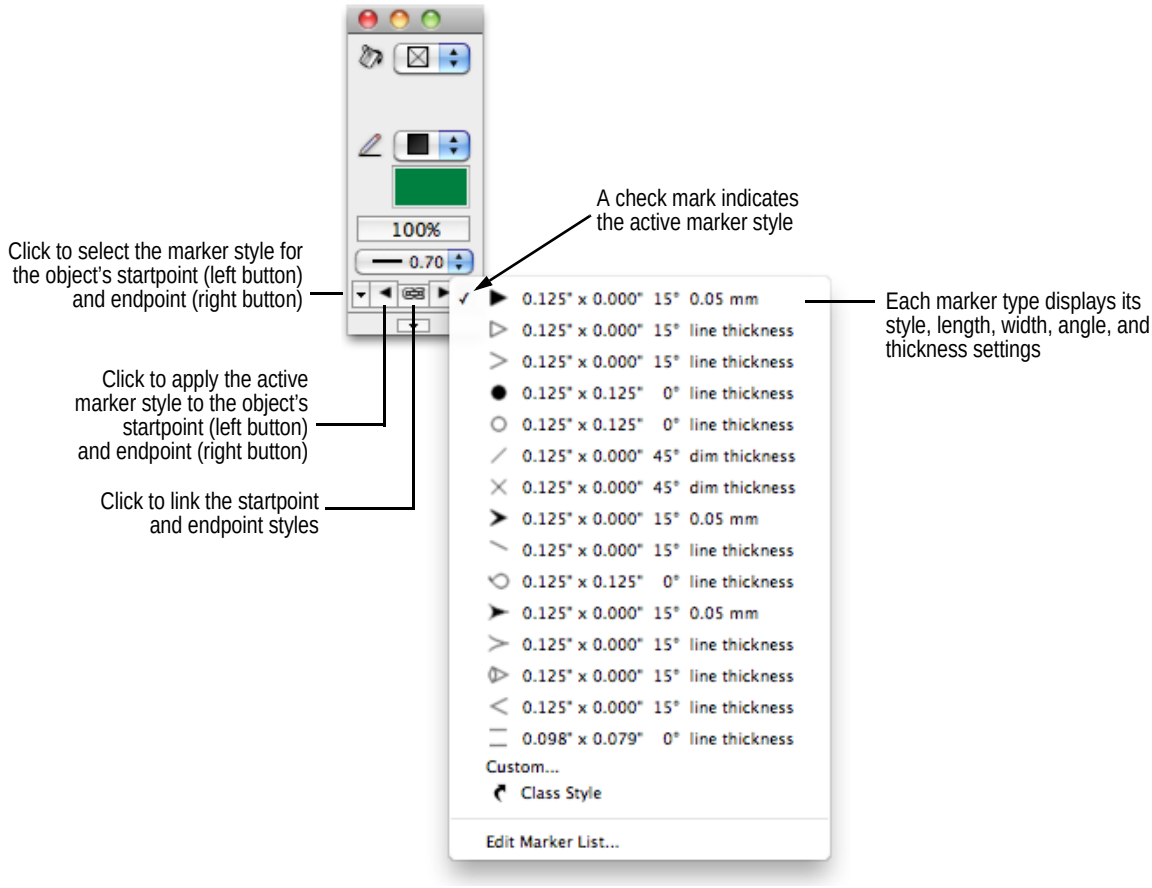
In the Attributes palette, click the **Line Start Marker Toggle** button, the **Line End Marker Toggle** button, or both buttons to specify the end(s) to receive the displayed marker. When a marker is applied to an object, the toggle button is highlighted. To always use the same style for both ends of the object, also click the **Marker Link Toggle** button; whenever the marker style of one end is changed, the other end's marker style automatically changes to match.

To switch to a different marker style, click **Line Startpoint Style** or **Line Endpoint Style**. Select a marker style from the list, or click **Edit Marker List** to create a new style. See "Editing the Marker List" on page 1103 for more information.

Select **Class Style** to use the marker attributes set by the object's class.

- If **Use at Creation** is enabled for the object's class, the object's marker style is automatically set by the class.
- If **Use at Creation** is disabled for the class, the object only takes on the class style when the **Class Style** option is selected.

See "Setting Class Attributes" on page 181.



Editing the Marker List

You can customize the set of markers available from the Line Startpoint Style and Line Endpoint Style settings on the Attributes palette. The edited markers are saved with your user preferences, so they are available from one Vectorworks session to the next.

To edit the marker list:

1. Select **Tools > Options > Edit Marker List**.

   The Edit Marker List dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Current Marker Types	Lists the marker types that are available for selection when drawing; change the list order by clicking and dragging in the # column
New	Creates a new marker type
Edit	Edits the currently selected marker type
Delete	Deletes the currently selected marker

2. Click **OK** to save the list of available marker types.

Creating or Editing Marker Types

To create or edit marker types:

1. From the Edit Marker List dialog box, click **New** to create a new marker type, or click **Edit** to change the currently selected marker type.

   The Edit Marker dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Preview	Displays a preview of the marker's appearance
General Options	Specifies the marker's shape and style
Root Type	Specifies the general shape of the marker
Fill	For closed root types, specifies the type of fill
Base	For triangular root types, specifies the shape of the base
Angle	For triangular and hexagonal root types, indicates the angle of the root
Length	Indicates the length of the marker, from base to tip
Width	Specifies the marker width, for marker types that have a width
Half Tick	For marker types except cone and lasso, displays half the marker on the indicated side
Tail	For markers which can be reversed, flips the marker direction to create a tail marker
Thickness Options	Specifies the marker pen thickness
Use Line Thickness	Uses the same thickness as that of the associated line, adjusting automatically along with any line thickness changes
Use Dimension Slash Thickness	Uses the same thickness as the dimension slash, set on the Dimensions tab of the document preferences (see "Dimension Preferences" on page 61)
Use Custom Thickness	Specifies a custom marker thickness and unit (mils, points, or millimeters)

2. Click **OK** to create or edit the marker.

   The new marker is listed at the top of the marker types list.

   Click here for a video tip on this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating or Editing Marker Types
Using Line Types
The Attributes Palette

Using Hatch Fills

Hatches can be applied to 2D planar and screen objects and walls (wall hatches are only visible in Top/Plan view).

There are two types of hatches: associative and non-associative. The appearance of both types of hatches are determined by hatch pattern definitions.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Associative Hatches

Associative hatch fills are resources; you can select them from the default content in the Attributes palette (see "Resource Libraries" on page 219), or you can create or import custom resources and apply them from the Resource Browser (see "Accessing Existing Resources" on page 229).

There are two ways to customize the hatch to suit the object to which it is applied; use either the **Fill Hatch Settings** button in the Attributes palette, or use the **Attribute Mapping** tool to move, rotate, or scale the hatch (see "Mapping Hatch and Tile Fills" on page 1123).

Associative hatches can be specified as a default class attribute (see "Setting Class Properties" on page 179).

Non-associative Hatches

Non-associative hatches are placed on objects or areas of the drawing with the **Hatch** command. They obtain most of their attributes from hatch pattern definitions, but they do not use the color definitions for the lines that make up the hatch. Non-associative hatches obtain the color definitions from the current default attributes. They do not use any background definitions, and therefore have no background color. A non-associative hatch is similar to a screen that displays over other objects; the spaces between the lines are empty, and display portions of any objects behind the hatch.

Non-associative hatches are placed inside an area that is defined by selected objects or lines. The start point of the hatch is set with the **Hatch** command. Unlike associative hatches, non-associative hatches can be moved to another area or object. However, unless the new area is the same shape and size as the previous one, the pattern will not fit.

Non-associative hatches are groups that are not associated with an object. They do not rotate with the object or act like a fill. They can be broken down into their individual elements with the **Ungroup** command. For a non-associative hatch to become part of an object, it must be grouped with the object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Hatches

Both associative and non-associative hatches must be defined. A hatch definition is a repetition of the elements in a series of lines in all directions from the beginning point. Specify where the hatch line begins (the **Start Point**), where it stops (the **Dash Factor**), where it begins to repeat (the **Repeat**), and the distance separating the line from a neighboring line (the **Offset**); the pattern is repeated in all directions.

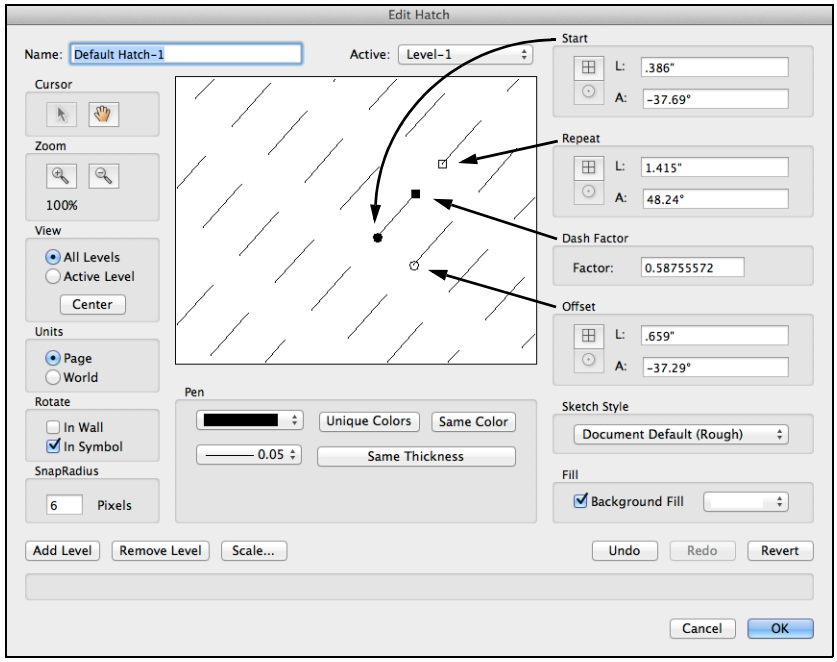
A hatch can consist of several levels, or layers, of pattern definitions. Each level is edited individually to create the overall hatch. (Hatch layers are named hatch levels in order to distinguish them from the layers in the drawing area.)

A new hatch can be created by editing an existing hatch; see "Editing Hatch Definitions" on page 1111.

To create a new hatch pattern definition:

1. Open the Resource Browser by selecting **Window > Palettes > Resource Browser**.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Hatch**.

   The Edit Hatch dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click to show/hide the parameters.

Parameter	Description
Name	Enter a name for the hatch; this name identifies the hatch in the Resource Browser and in the Select Hatch dialog box (**Modify > Hatch**)
Active	Select the active hatch level from the list. A level can also be selected by clicking on one of its pattern lines, or by using the keyboard right and left arrow keys. Levels are numbered in the order of creation.
Cursor	Switches between the pointer and pan functions. The pointer adjusts the four control handles in the hatch window. The pan moves the elements of the hatch around the preview window.
Zoom	Zooms in and out by a factor of two. The center of the view remains at its current coordinates. The current zoom ratio is also displayed. The initial zoom setting when the dialog box opens corresponds to the best setting for editing Level 1.
View	Select whether to view **All Levels** or only the **Active Level**; click **Center** to center the start point of the active level
Units	Switches between **Page** and **World** settings for the hatch. **Page** is absolute in screen coordinates, where one inch in the hatch equals one inch on the screen (at 100% zoom). **World** sets the hatch to use the unit values for the layer where the hatch will be placed; for example, one inch in the hatch equals one inch in the drawing area as defined by the rulers (this may not equal one inch on the screen depending on the unit setting). When switching between **Page** and **World**, the hatch settings are adjusted for all levels to preserve the hatch appearance.
Rotate	Select whether to rotate the hatch to match the orientation of any symbols or walls where it will be placed
Snap Radius	Set the snap radius for snapping to the end points and mid points of pattern lines when moving a line by dragging

Parameter	Description
Pen color	Sets the color for the active level hatch lines; click the color box to select a color. Click **Unique Colors** to set a unique color for each level; click **Same Color** to apply the active level color selection to all levels.
Pen line weight	Sets the hatch line weight for the current level. Click the line to select a weight from the list. Create a custom weight by selecting **Set Thickness**. The **Set Thickness** dialog box opens. Enter the **Thickness Value** and **Thickness Units**. Click **Same Thickness** to apply the active level line thickness to all levels.
Start Point	Controls the location of the start of the first hatch line in relation to the hatch origin for the active level. The default mode uses polar coordinates (L = distance, A = angle). Enter values or move the Start Point handle in the preview window.
Repeat	Sets the distance, for the active level, between the beginning of a segment and the beginning of the next collinear segment. **Repeat** directly relates to the **Dash Factor**, with the two determining if the line is dashed or solid, as well as the length of the dashes and line segments. The default mode uses polar coordinates (L = distance, A = angle). Enter values or move the Repeat handle in the preview window.
Dash Factor	Represents, for the active level, the percentage of the distance between the **Start Point** and the **Repeat** that is shown as a line. Set to 1 to create a solid line. Enter a value or move the Dash Factor handle in the preview window.
Offset	Determines the distance and direction between lines for the active level. The values entered are relative to the **Start Point**. The default mode uses polar coordinates (L = distance, A = angle). Enter values or move the Offset handle in the preview window.
Sketch Style	For Vectorworks Design Series products, specifies a hatch sketch style; see “Applying Sketch Styles to Hatches” on page 1501
Background Fill	Select to use a background color and click the color box to select a color; background color applies only to associative hatches
Add/Remove Level	Click **Add Level** to create an offset duplicate of the active level. Alternatively, create a duplicate level by clicking and dragging the **Start Point** handle with the Option (Mac) or Alt (Windows) key. Click **Remove Level** to remove the active level.
Scale	Opens the Scale Hatch Definition dialog box. Enter a **Scaling Factor** to change the hatch definitions for the active level (except for the Dash Factor value, which remains unchanged). Select **Apply to All Levels** to change the scale factor for all levels.
Undo	The last five actions can be undone by clicking **Undo**
Redo	Click to redo the last action that was undone. Must be clicked directly after an action was undone. Appears dimmed when there are no actions to undo.
Revert	Returns the hatch definition to its status at the time the Edit Hatch dialog box was opened

The **Start Point**, **Repeat**, **Dash Factor**, and **Offset** functions correspond to the four control handles in the preview window. When a handle is moved, the corresponding function’s values change to reflect the move. The Shift key constrains the drag when using the control handles, affecting each of the four functions differently.

For the **Start Point**, **Repeat**, and **Offset** parameters, enter values according to either the polar coordinate system or the Cartesian system. The text box labels for these four controls change to correspond to the selection of Cartesian or polar.

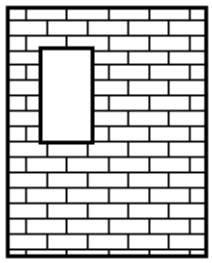
Coordinate System	Description
Cartesian	Cartesian coordinates are the same as those used in the drawing area, with positive and negative X and Y axes
Polar	Defines the different hatch line segments by length and angle values. In polar mode, angles are represented as positive or negative values from 0 to 180. Values from 180 to 359 are automatically converted to negative. 0 is at the 3 o'clock position.

4. Click **OK**. The new hatch definition is saved under the specified name.

Example: Creating a Brick Hatch

The following Edit Hatch values demonstrate how to create a brick pattern. The hatch requires two levels. The bricks will be 8"x 3" and the first set of lines will be the vertical components of the bricks. This procedure creates a hatch that scales appropriately when it is placed into a drawing.

All values are entered in World units and Cartesian mode.



Click to show/hide the parameters.

Parameter	Level 1 Values	Level 2 Values
Start Point	X = 0, Y = 0	X = 0, Y = 0
Repeat	X = 0, Y = 6	X = 2, Y = 0
Dash Factor	0.5	1
Offset	X = 4, Y = 3	X = 0, Y = 3

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Associative Hatches
Applying Non-Associative Hatches
Editing Hatch Definitions

Applying Associative Hatches

Apply hatches to specific objects using either the Attributes palette or the Resource Browser. Some objects have the ability to apply hatch settings while setting the object attributes.

> Attributes can also be applied by class settings (see "Setting Class Attributes" on page 181) or transferred with the **Eyedropper** tool (see "Transferring Attributes" on page 1095).

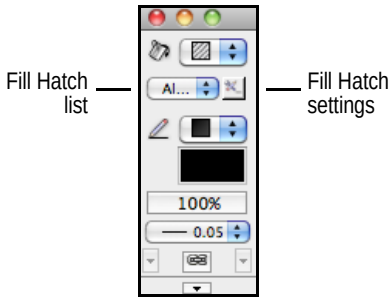
Applying a Hatch from the Attributes Palette

To apply an associative hatch from the Attributes palette and specify the hatch settings:

1. Select the object(s), and then select **Hatch** from the Attributes palette fill list.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

If no hatches are defined, and default content is not enabled in Vectorworks preferences, you are prompted to add a default hatch definition.

2. To change to a different hatch, click on the hatch list and select a different hatch from the list of default content or the current file's content (see "Resource Libraries" on page 219).



3. To specify the settings for the current hatch, click the **Fill Hatch Settings** button next to the Fill Hatch list.

   The Hatch Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Hatch selector	Select a hatch resource from either the default content or the current file's content
Use Local Mapping	Select to enable entry in the mapping fields, to customize the fill for this object; deselect to return to the default fill mapping. The **Attribute Mapping** tool is an alternate way to create a local mapping; if an object's fill is edited with the **Attribute Mapping** tool, the changes are reflected on this dialog box. Conversely, edits made on this dialog box are reflected in the position, size, and rotation of the mapping tool's editing object. By default, a hatch fill does not rotate with the filled object; select **Use Local Mapping** to keep the fill's orientation relative to the object.
X/Y Offset	Indicates the fill starting point coordinates relative to the center of the selection bounding box (in the file's current units)
I/J Length	Specifies the distance in the I/J direction for a single instance of the fill (in the file's current units). To maintain the aspect ratio of the fill when one length is changed, click the link button; the other length value changes automatically.
Rotation	Specifies the angle of the fill
Flip Horizontally/ Flip Vertically	Select to flip the fill orientation horizontally, vertically, or both

4. If desired, click **Preview** to view the results of the fill settings.
5. Click **OK** to apply the fill settings.

Applying a Hatch from the Resource Browser

To apply an associative hatch from the Resource Browser:

1. Select the object(s).

2. From the Resource Browser, select the resource to be applied. Right-click (Windows) or Ctrl-click (Mac) and select **Apply** from the context menu. (Alternatively, double-click the resource to apply it to the selection or drag the resource onto an object.)

   The fill settings can be edited from the Attributes palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Transferring Attributes
Using Hatch Fills
Defining Hatches
Applying Non-Associative Hatches
Editing Hatch Definitions
Mapping Hatch and Tile Fills

Applying Non-Associative Hatches

To apply a non-associative hatch:

1. Select the object(s).
2. Select **Modify > Hatch**.

   The Hatches dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Hatches list	Lists the available hatches from the default content, the current file's content, or from a referenced file (referenced hatches display in italics)
Hatch preview	Displays a preview of the selected hatch at 100% zoom
New	Opens the Edit Hatch dialog box, for creating a new hatch
Edit	Opens the Edit Hatch dialog box, for editing or renaming an existing hatch
Duplicate	Creates a copy of the selected hatch (button is disabled if referenced hatches are present)
Delete	Deletes the selected hatch

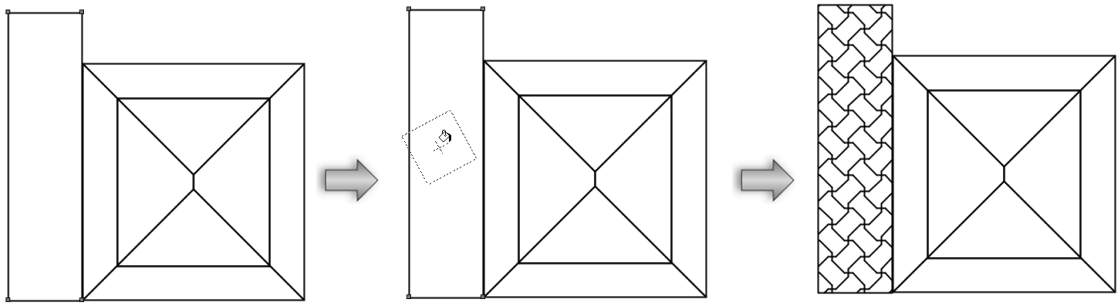
3. Select the hatch to apply from the **Hatches** list.

   A preview of the hatch is displayed.
4. Click **OK** to return to the drawing area.

   The cursor changes to a paint bucket.
5. Position the paint bucket on the object where the hatch should begin. Click to specify the hatch origin. Drag to specify the hatch orientation and click again.

   The hatch fills the space from the paint bucket's location to the boundary created by the object.

   When a hatch is being applied to an object, the tip of the paint from the paint bucket marks the hatch origin. The hatch start point within the object can be precisely specified.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Select the object

After selecting the hatch, click the cursor at the desired insertion point. The hatch can be rotated during insertion.

Click again to place the hatch once it is oriented

Using Hatch Fills
Defining Hatches
Applying Associative Hatches
Editing Hatch Definitions

Editing Hatch Definitions

Changes to a hatch definition affect all associative instances of that hatch in the drawing file. Non-associative hatches are not affected.

To edit a hatch definition:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.

2. Select the desired hatch. Right-click (Windows) or Ctrl-click (Mac), and select **Edit** from the context menu.

   Alternatively, select a drawing object, select **Modify > Hatch**, and click **Edit** for the selected hatch in the Hatches dialog box.

   The Edit Hatch dialog box opens.

3. Edit the hatch as described in “Defining Hatches” on page 1105.

4. Click **OK** to return to the drawing area.

   Any associative instance of the edited fill in the drawing file changes according to the new definition.

Defining Hatches
Applying Associative Hatches
Applying Non-Associative Hatches
Mapping Hatch and Tile Fills

Using Tile Fills

Tile fills are resources; you can select them from the default content in the Attributes palette (see “Resource Libraries” on page 219), or you can create or import custom resources and apply them from the Resource Browser (see “Accessing Existing Resources” on page 229).

There are two ways to customize the fill to suit the object to which it is applied; use either the **Fill Tile Settings** button in the Attributes palette, or use the **Attribute Mapping** tool to move, rotate, or scale the tile.

Tile fills can be applied to any type of 2D planar or screen object that accepts a fill, including walls (wall tiles only display in Top/Plan view), text boxes, worksheet backgrounds, and plug-in objects that include a 2D object.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Tiles

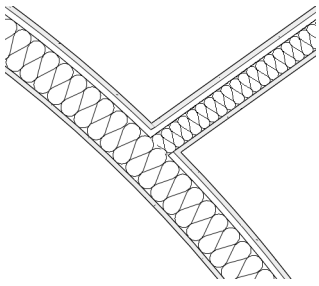
A tile is a set of 2D geometric elements that repeats in all directions from a center point. The geometric elements can have color or fill (but not a tile fill), and the tile can also have a background color.

To create a tile definition:

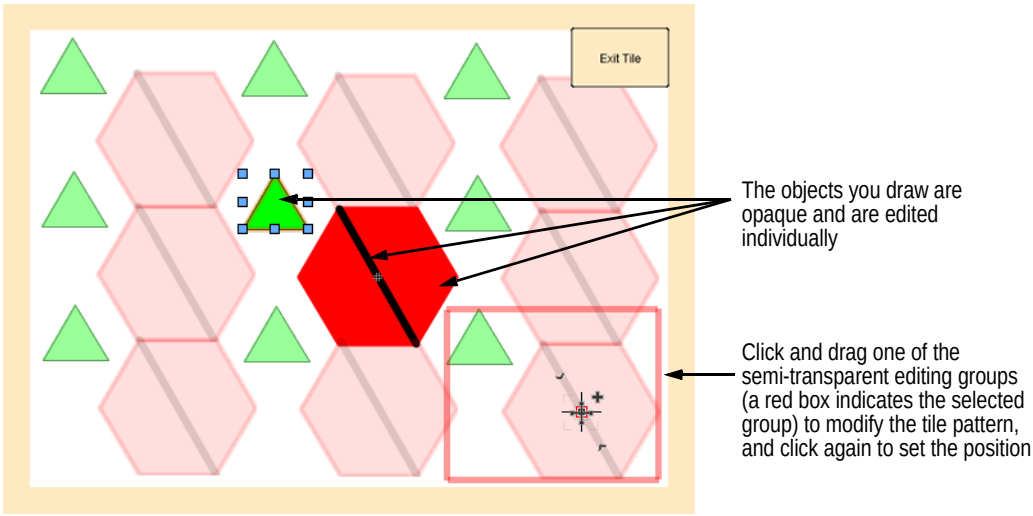
1. Open the Resource Browser by selecting **Window > Palettes > Resource Browser**.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Tile**.

   The New Tile dialog box opens.

Click to show/hide the parameters.

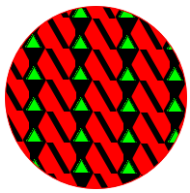
Parameter	Description
Name	Specifies a name for the tile resource; this name is displayed in the Resource Browser and Attributes palette
Units	Specifies the units for the tile when it is used in a drawing: • **Page** sets the tile size according to the page dimensions. When applied, the tile automatically scales relative to the page environment. This is useful for annotation objects that should always remain the same size on the "page," regardless of the layer scale. • **World** sets the tile size according to world dimensions; its size varies depending on the layer scale.
Rotate	Rotates the tile fill according to the axis of any wall or symbol to which it is applied
In Wall	Matches the tile rotation to the wall axis; select **Fit to Wall** to scale the tile fill to match the thickness of a wall, wall component, or slab component. Enable the **Fit to Wall** option for insulation fills in walls, and wall or slab components. 
In Symbol	Matches the tile rotation to the symbol axis
Background Fill	To have a background fill behind the tile geometry, select the **Background Fill** option and also specify a color.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

4. Click **OK** to open a tile editing window. Draw the tile geometry with the desired colors and fills. When you create an object, eight partially transparent repetitions of the object display around it. You can add multiple objects to the pattern.
5. To adjust the tile pattern and spacing, click one of the repetitions and drag it as needed; click again to set the new position. To hide the tile repetitions during editing, Ctrl-click (Mac) or right-click (Windows) on an empty area of the editing window, and deselect the **Display Tile Repetitions** option on the context menu.



6. When you are done drawing the tile, click **Exit Tile**. The tile resource is saved with the specified name.

   Tiles are saved in Vectorworks drawing files. If the file is not saved and the tile is not default content, the tile is lost when you exit the program.



Applying Tiles
Editing Tile Definitions
Mapping Hatch and Tile Fills

Applying Tiles

Apply tiles to specific objects using either the Attributes palette or the Resource Browser. Some objects have the ability to apply tile settings while setting the object attributes.

> Attributes can also be applied by class settings (see “Setting Class Attributes” on page 181) or transferred with the **Eyedropper** tool.

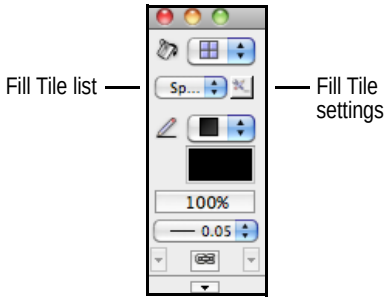
Applying a Tile from the Attributes Palette

To apply a tile from the Attributes palette and specify the tile settings:

1. Select the object(s), and then select **Tile** from the Attributes palette fill list.

   If no tiles are defined, and default content is not enabled in Vectorworks preferences, you are prompted to add a default tile definition.

2. To change to a different tile, click on the tile list and select a different tile from the list of default content or the current file's content (see "Resource Libraries" on page 219).



3. To specify the settings for the current tile fill, click the **Fill Tile Settings** button next to the Fill Tile list.

   The Tile Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Tile selector	Select a tile resource from either the default content or the current file's content
Use Local Mapping	Select to enable entry in the mapping fields, to customize the fill for this object; deselect to return to the default mapping. The **Attribute Mapping** tool is an alternate way to create a local mapping; if an object's fill is edited with the **Attribute Mapping** tool, the changes are reflected on this dialog box. Conversely, edits made on this dialog box are reflected in the position, size, and rotation of the mapping tool's editing object. By default, a tile fill does not rotate with the filled object; select **Use Local Mapping** to keep the fill's orientation relative to the object.
X/Y Offset	Indicates the fill starting point coordinates relative to the center of the selection bounding box (in the file's current units)
I/J Length	Specifies the distance in the I/J direction for a single instance of the fill (in the file's current units). To maintain the aspect ratio of the fill when one length is changed, click the link button; the other length value changes automatically.
Rotation	Specifies the angle of the fill
Flip Horizontally/ Flip Vertically	Select to flip the fill orientation horizontally, vertically, or both

4. If desired, click **Preview** to view the results of the fill settings.
5. Click **OK** to apply the fill settings.

Applying a Tile from the Resource Browser

To apply a tile from the Resource Browser:

1. Select the object(s).
2. From the Resource Browser, select the resource to be applied. Right-click (Windows) or Ctrl-click (Mac) and select **Apply** from the context menu. (Alternatively, double-click the resource to apply it to the selection or drag the resource onto an object.)

   The fill settings can be edited from the Attributes palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Transferring Attributes
Defining Tiles
Editing Tile Definitions
Mapping Hatch and Tile Fills

Editing Tile Definitions

Changes to a tile definition affect all instances of that tile in the drawing file.

To edit a tile definition:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.
2. Select the desired tile. Right-click (Windows) or Ctrl-click (Mac), and select **Edit** from the context menu.
3. The Edit Tile dialog box opens; select whether to edit the Geometry or Settings.
   - If you edit the settings, the Edit Tile Settings dialog box opens next.
   - If you edit the geometry, the tile editing window opens next to allow editing of the tile components.
4. Adjust the tile definition as described in “Defining Tiles” on page 1112.
5. Any instance of the edited fill in the drawing file changes according to the new definition.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Tiles
Applying Tiles
Mapping Hatch and Tile Fills

Using Gradient Fills

Gradient fills are resources; you can select them from the default content in the Attributes palette (see “Resource Libraries” on page 219), or you can create or import custom resources and apply them from the Resource Browser (see “Accessing Existing Resources” on page 229).

Gradients can be applied to any type of 2D planar or screen object that accepts a fill, including walls (wall gradients only display in Top/Plan view), text boxes, worksheet backgrounds, and plug-in objects that include a 2D object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Gradients
Applying Gradients
Editing Gradient Definitions
Mapping Gradient Fills
Fill Attributes
Transferring Attributes

Defining Gradients

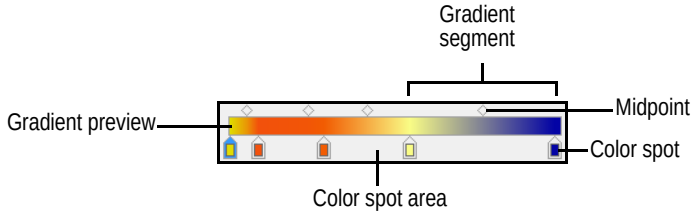
A gradient is a set of two or more colors that blend smoothly into one another, or into a different opacity. Gradients are defined and stored in the Resource Browser.

To create a gradient definition:

1. Open the Resource Browser by selecting **Window > Palettes > Resource Browser**.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Gradient**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Gradient Editor dialog box opens. Specify a name for the gradient resource, and select the segment starting and ending colors and their opacity settings. Gradients can consist of several segments and more than two colors. To create a gradient with more than two colors, click in the color spot area. This adds a color spot and midpoint to the gradient. Each color spot has an opacity value.

Specify the location of a selected color spot or midpoint by dragging it into position, or entering its position in the **Position** field.



Click to show/hide the parameters.

Parameter	Description
Name	Specifies a name for the gradient resource; this name is displayed in the Resource Browser and Attributes palette
Midpoint	Point between two colors where each color/opacity is of equal intensity; a midpoint is associated with the color spot to its left. By default, the midpoint is placed equidistant between two color spots; but it can be dragged to another location.
Gradient Preview	Displays the gradient in a preview bar
Color Spot	Specifies the starting or ending color and opacity of each gradient segment. To specify the color, double-click on a color spot, or select a color spot and click **Color**. To add a new gradient segment, click in an empty location in the color spot area. A new color spot and midpoint are created. Drag a color spot to a new location in the color spot area; its associated color/opacity is retained. Remove a color spot and its associated midpoint by selecting the color spot dragging it from the color spot area.
Color	Specifies the color of the selected color spot; click to select the color
Opacity	For the selected color spot, drag the slider to the left to decrease the opacity, or enter an opacity percentage (0 – 100) to the right of the slider. A gray checkerboard pattern is revealed in the **Gradient Preview** as the opacity decreases. To create an opacity gradient for a single color, specify the same color for all color spots and set the color spots to different opacities.
Position	Indicates the position (0.0 – 1.0) of the selected color spot or midpoint; the midpoint position is relative to its location between two adjacent color spots

4. Click **OK** to save the gradient resource with the specified name.

Gradients are saved in Vectorworks drawing files. If the file is not saved and the gradient is not default content, the gradient is lost when you exit the program.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Gradients
Editing Gradient Definitions
Mapping Gradient Fills
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

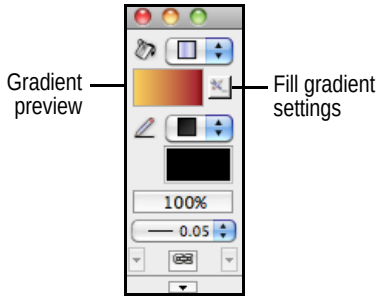
Applying Gradients

Gradient settings are specified through the Attributes palette and applied to 2D planar and screen objects through the Resource Browser or the Attributes palette. In addition, a gradient fill can be specified as a default class attribute (see “Setting Class Properties” on page 179). Some objects have the ability to apply gradient settings while setting the object attributes.

Applying a Gradient from the Attributes Palette

To apply a gradient resource from the Attributes palette and specify the gradient settings:

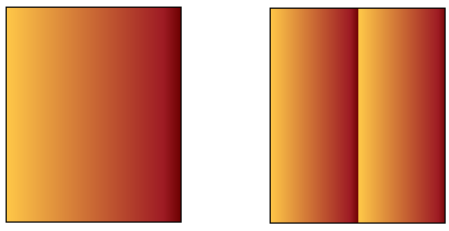
1. Select the object(s), and then select **Gradient** from the Attributes palette fill list. To change to a different gradient, click on the gradient preview and select a different gradient from the list of default content or the current file’s content (see “Resource Libraries” on page 219).

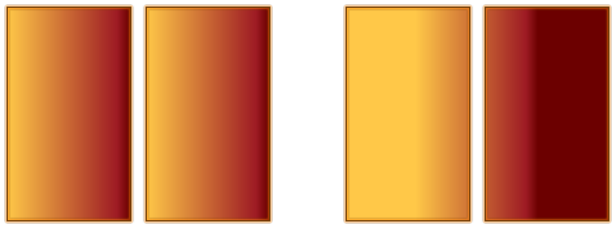
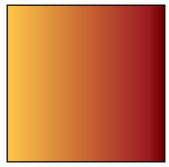
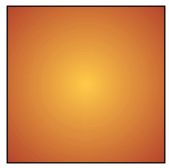
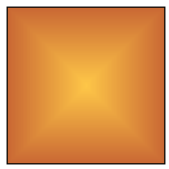
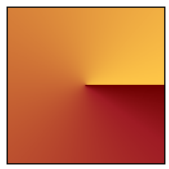


2. Specify the settings for the current gradient by clicking the **Fill Gradient Settings** button next to the gradient preview box.

   The Gradient Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Gradient selector	Select a gradient resource from either the default content or the current file’s content
Repeat	Select to repeat the gradient segment(s) over the object; deselect to apply a single instance of the gradient segment(s) to the object 

Parameter	Description
Apply To	Select **Each Object** to apply the gradient to each selected object individually; choose **Selection** to apply the gradient across the selected objects, spanning the objects.  Each Object Selection When applying a gradient across several selected objects, group the objects to retain the effect.
Type	Select the gradient type from the list
Linear	Applies the gradient to the selection with linear geometry 
Radial	Applies the gradient to the selection with circular geometry 
Rectangular	Applies the gradient to the selection with rectangular geometry 
Angular	Applies the gradient to the selection in a counterclockwise direction from the specified starting point 
X/Y Offset	Indicates the gradient starting point coordinates relative to the center of the selection bounding box (in the file's current units)
Length	Specifies the length of a single gradient segment (in the file's current units)
Rotation	Specifies the rotation of the gradient fill
Reverse	Select to draw the colors in reverse order from the order specified in the Gradient Editor dialog box

3. If desired, click **Preview** to view the results of the gradient settings.
4. Click **OK** to apply the gradient settings.

Applying a Gradient from the Resource Browser

To apply a gradient resource from the Resource Browser:

1. Select the object(s).
2. From the Resource Browser, select the gradient to be applied. Right-click (Windows) or Ctrl-click (Mac), and select **Apply** from the context menu. (Alternatively, double-click the gradient resource to apply it to the selection or drag the gradient resource onto an object.)

   The fill settings can be edited from the Attributes palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Transferring Attributes
Defining Gradients
Editing Gradient Definitions
Mapping Gradient Fills

Editing Gradient Definitions

Changes to a gradient definition affect all instances of that gradient in the drawing file.

To edit a gradient definition:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.
2. Select the desired gradient. Right-click (Windows) or Ctrl-click (Mac), and select **Edit** from the context menu.
3. Adjust the gradient definition as described in “Defining Gradients” on page 1115.
4. Click **OK** to return to the drawing area.

   Any instance of the edited fill in the drawing file changes according to the new definition.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Gradients
Applying Gradients
Mapping Gradient Fills

Using Image Fills

Image fill attributes are resources; you can select them from the default content in the Attributes palette (see “Resource Libraries” on page 219), or you can create or import custom resources and apply them from the Resource Browser (see “Accessing Existing Resources” on page 229).

Images can be applied to any type of 2D planar or screen object that accepts a fill, including walls (wall images only display in Top/Plan view), text boxes, worksheet backgrounds, and plug-in objects that include a 2D object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Image Resources
Applying Image Resources
Mapping Image Fills
Fill Attributes
Transferring Attributes
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Image Resources

Images resources are imported and stored in the Resource Browser.

You can also import an image directly (not for use as a resource). For example, you might import a sketch or a logo graphic into a drawing. See “Importing an Image File” on page 1680 for details.

To import an image for use as an image resource:

1. Open the Resource Browser by selecting **Window > Palettes > Resource Browser**.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Image**.
4. If a resource with an image is already present in the file, the Choose Image dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Import an Image File	Imports a new image; click **OK** and proceed to Step 5.
Reuse an Image from Another Resource	Reuses a previously imported image; select the resource that contains the image. Click **OK** and proceed to Step 6.

5. The Open dialog box is displayed. Select the image file to use as an image resource, and click **Open**.
6. If the image is in JPEG format, it is imported immediately. For images not in JPEG format, the Image Import Options dialog box opens. Specify the imported image options.

   The current image information is displayed at the top, along with the image’s uncompressed size. Two compression methods are available; depending on the graphic, one of the methods may be more suitable. The compression method which produces the smallest file size is selected by default. Select the desired balance between compression and detail display. The resulting file size for each type of compression is displayed to help with the selection.

   If the selected option results in a file size larger than the uncompressed size, the image is imported uncompressed.

Compression Method	Description
JPEG	Provides a high amount of compression, resulting in the smallest Vectorworks file size. However, fine detail may be obscured. JPEG compression is most suitable for photographic images.
PNG	Provides a moderate amount of compression, while preserving image details; an image compressed as a PNG can also be imported as a 1-bit monochrome image by selecting **Import as Black and White**
Referencing options (Vectorworks Design Series required)	Images can be referenced; see “Referencing Resources” on page 213

7. Click **OK** to import the image with the selected compression type. The image resource is saved, by default, with the name of the original image file.

Image resources imported into the file that are not in JPEG format can be compressed by JPEG with the **Compress Images** command. See “Compressing Images” on page 1822.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Image Resources
Mapping Image Fills
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

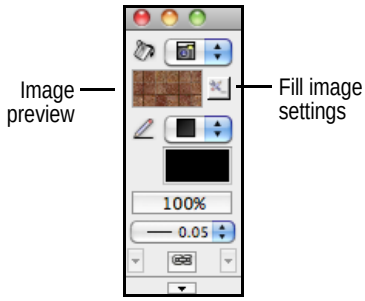
Applying Image Resources

Image settings are specified through the Attributes palette and applied to a 2D object through the Resource Browser or the Attributes palette. In addition, an image fill can be specified as a default class attribute (see “Setting Class Properties” on page 179). Some objects have the ability to apply image settings while setting the object attributes.

Applying an Image from the Attributes Palette

To apply an image resource from the Attributes palette and specify the image settings:

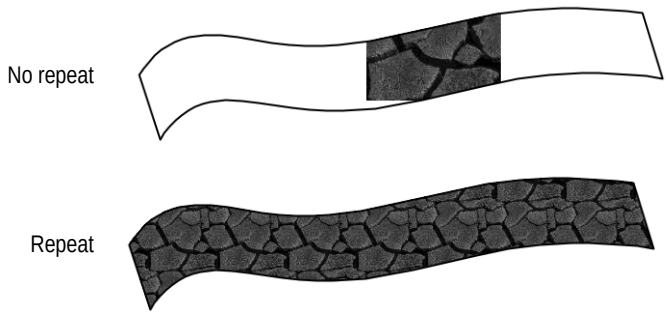
1. Select the object(s), and then select **Image** from the Attributes palette fill list. To change to a different image, click on the image preview bar and select a different image from the list of default content or the current file’s content (see “Resource Libraries” on page 219).

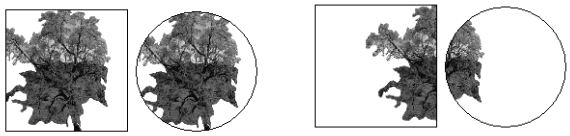


2. Specify the settings for the current image by clicking the **Fill Image Settings** button next to the image preview box.

   The Image Settings dialog box opens.

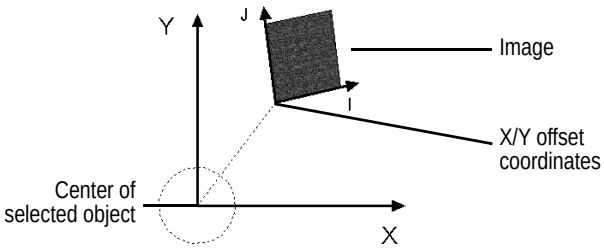
Click to show/hide the parameters.

Parameter	Description
Image selector	Select an image resource from either the default content or the current file’s content
Repeat	Select to repeat the image over the object; deselect to apply a single instance of the image to the object 

Parameter	Description
Apply To	Select **Each Object** to apply the image to each selected object individually; choose **Selection** to apply the image across the selected objects, spanning the objects.  Each Object Selection When applying an image across several selected objects, group the objects to maintain the image.
X/Y Offset	Indicates the image starting point coordinates relative to the center of the selection bounding box (in the file's current units)
I/J Length	Specifies the distance in the I/J direction for a single instance of the image fill (in the file's current units). To maintain the aspect ratio of the image when one length is changed, click the link button; the other length value changes automatically.
Rotation	Specifies the angle of the image fill
Flip Horizontally/ Flip Vertically	Select to flip the image orientation horizontally, vertically, or both

3. If desired, click **Preview** to view the results of the image settings.
4. Click **OK** to apply the image settings.

   The X and Y axes are file-based, relative to the center of the selected object(s). The I and J axes are image-based.



Applying an Image from the Resource Browser

To apply an image resource from the Resource Browser:

1. Select the object(s).
2. From the Resource Browser, select the image to be applied. Right-click (Windows) or Ctrl-click (Mac), and select **Apply** from the context menu. (Alternatively, double-click the image resource to apply it to the selection or drag the image resource onto an object.)

   The image settings can be edited from the Attributes palette.

Transferring Attributes
Creating Image Resources
Mapping Image Fills

Mapping Fills with the Attribute Mapping Tool

Use the **Attribute Mapping** tool to edit the size, position, and angle of an associative hatch, tile, gradient, or image fill that has been applied to an object.

If you are working in 3D and have the Renderworks product installed, you can use the **Attribute Mapping** tool to map textures directly in the drawing window. See "Direct Texture Mapping" on page 1535.

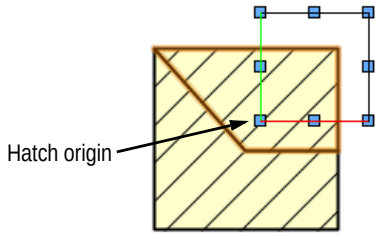
Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

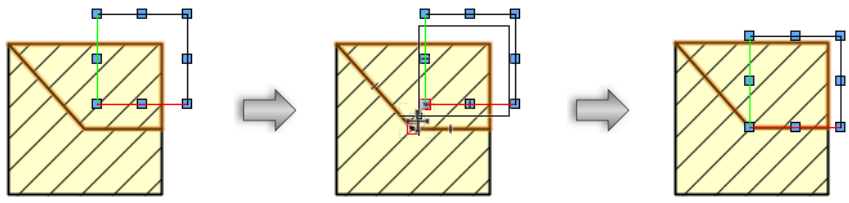
Mapping Hatch and Tile Fills

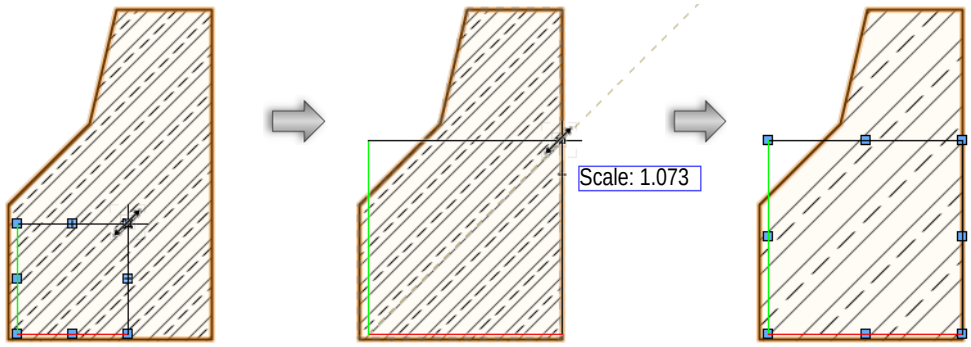
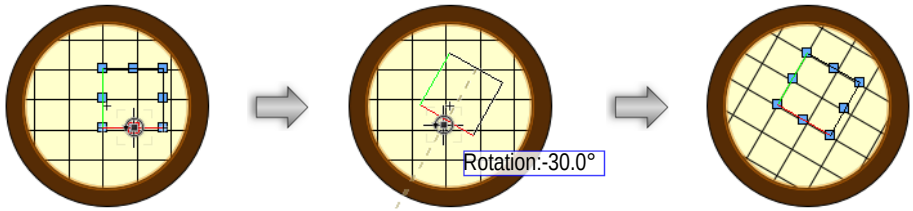
To modify an object's tile or associative hatch fill:

1. Click the **Attribute Mapping** tool from the Basic palette or Visualization tool set.
2. Click on an object containing a tile or associative hatch fill. Only one object at a time can be edited.
3. An editing object with eight handles is placed over the fill. The fill origin is at the lower left corner of the editing object, where the red and green lines meet.



4. Use the editing object to set the fill location, scale, and rotation angle. The fill can also be nudged.

Action	Description
Edit the fill location	Click in the editing object (anywhere except on a selection handle) and drag it to the desired location. Click to set. To precisely locate the origin, click close to the lower left handle, and move the editing object as needed. 
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Action	Description
Edit the fill scale	Click on one of the corner selection handles, and drag to set the editing object to a new scale (or press the Tab key and enter a **Scale** value in the Data bar). Click to set. 
Edit the fill rotation	Click on one of the middle selection handles, and drag to rotate the editing object to the desired angle (or press the Tab key and enter a **Rotation** value in the Data bar); click to set the rotation. • By default, the editing object rotates about the handle opposite the handle that was clicked on. • To rotate about the fill origin, press the Shift key while rotating. • To rotate about the center of the editing object, press the Ctrl key (Windows) or Command key (Mac) while rotating. 

5. In the Attributes palette, the name of the fill is highlighted in blue, and "(local map)" is appended to the name.

Creating a New Resource from a Locally Mapped Hatch or Tile

A hatch or tile that has been edited with the **Attribute Mapping** tool is considered to be locally mapped. The edited hatch or tile can be saved as a new hatch or tile resource, retaining the mapping edits and allowing them to be applied to other objects.

To create a hatch or tile from a locally mapped associative hatch or tile:

1. Right-click (Windows) or Ctrl-click (Mac) on an object with a locally mapped tile or associative hatch, and select **New Tile From Locally Mapped Tile** or **New Hatch From Locally Mapped Hatch** from the context menu.
2. Provide a name for the new hatch or tile resource.
3. The new resource appears in the Resource Browser. The new hatch or tile definition has already been applied to the object, which is no longer considered to be locally mapped.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

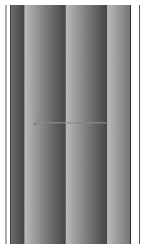
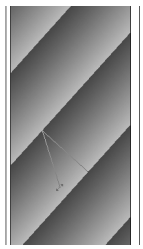
Defining Hatches
Defining Tiles
Fill Attributes
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Mapping Gradient Fills

To modify an object's gradient fill:

1. Click the **Attribute Mapping** tool from the Basic palette or Visualization tool set.
2. Click on an object containing a gradient fill. Only one object at a time can be edited.
3. An editing object with two handles is placed over the gradient.

4. Use the editing object to set the gradient origin location, i-axis length, and rotation angle. Hold down the Shift key to constrain the editing object line; the fill can also be nudged.

Action	Description
Edit the location of the fill (gradient origin)	Click and drag the entire editing object, moving it to the desired location. Click to set. 
Edit the gradient i-axis location	Click on a handle at the end of the editing object, and drag to set the editing object to a new length. Click to set. 
Edit the gradient rotation	Click on a handle at the end of the editing object, and move to create a fulcrum line; click to set the rotation. 

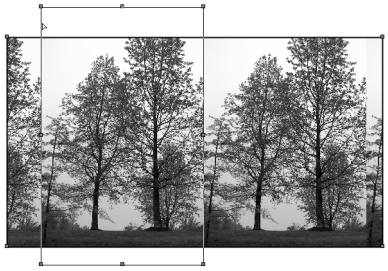
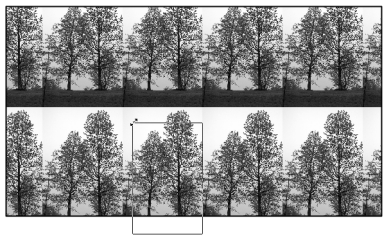
Defining Gradients
Fill Attributes

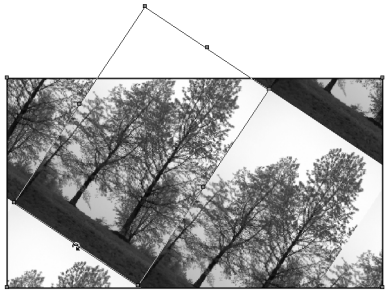
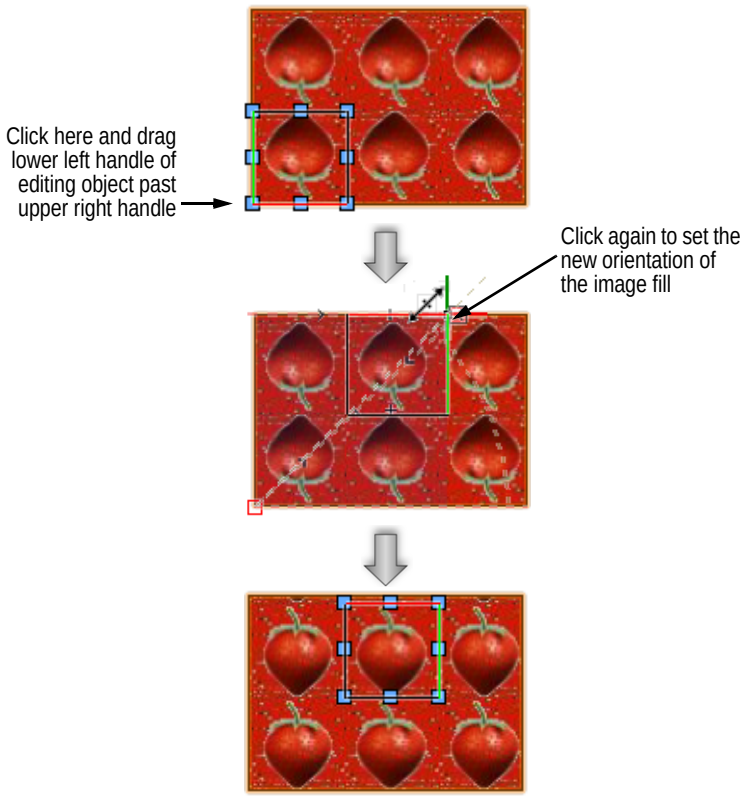
Mapping Image Fills

To modify an object's image fill:

1. Click the **Attribute Mapping** tool from the Basic palette or the Visualization tool set.
2. Click on an object containing an image fill. Only one object at a time can be edited.
3. An editing object with eight handles is placed over the image.

4. Use the editing object to set the image origin location, i-axis and j-axis lengths, and rotation angle. The fill can also be nudged.

Action	Description
Edit the location of the fill (image origin)	Click and drag the entire editing object, moving it to the desired location. 
Edit the image i-axis and j-axis location	Click on a corner handle (the resize cursor displays) and drag the handle to the new size; holding the Shift key during this operation constrains the image aspect ratio. Click to set. 

Action	Description
Edit the image rotation	Click on a side handle (the rotate cursor displays) and drag the handle to the new rotation angle (or press the Tab key and enter a **Rotation** value in the Data bar). Click to set. Press the Ctrl key (Windows) or Command key (Mac) while rotating to rotate about the image center. 
Flip the image horizontally or vertically	Click and drag a corner of the editing object past another corner of the editing object; click again to redraw the image within the new drawing area. Drag horizontally or vertically to flip the image in that direction; drag diagonally to flip the image both horizontally and vertically. 

Options in the Image Settings dialog box (accessed from the Attributes palette) also affect image attribute mapping. When the link button between the **I Length** and **J Length** fields is selected, the image cannot be flipped or mirrored by dragging the editing object's handles vertically or horizontally; only diagonal dragging is allowed. Use the **Flip Horizontally** or **Flip Vertically** options in this dialog box to change the fill image orientation without moving or resizing the image.

Creating Image Resources

Fill Attributes

Using Line Types

Line types are resources; you can select them from the default content in the Attributes palette (see "Resource Libraries" on page 219), or you can create or import custom resources and apply them from the Resource Browser (see "Accessing Existing Resources" on page 229).

From the Attributes palette, you can customize the line type to suit the object to which it is applied. Use the **Line Type Settings** button to adjust the line type scale or orientation, and adjust the **Pen Color** and **Line Thickness** as needed.

Line types can be applied to any type of 2D planar or screen object that has lines, including walls and plug-in objects.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Line Types
Applying Line Types
Editing Line Type Definitions

Defining Line Types

A line type is a set of 2D geometric elements that repeats along a line in both directions from a center point. The geometric elements can be a simple dash definition, or more complex 2D shapes with fill.

The color and thickness of the lines in a line type are not part of the resource definition. Once a line type is applied to an object, use the Attributes palette to set the color and thickness of the lines for that object. This allows more flexibility, so that you can have multiple objects that use the same line type, but use different colors, for example.

To create a line type definition:

1. Open the Resource Browser by selecting **Window > Palettes > Resource Browser**.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Line Type**.

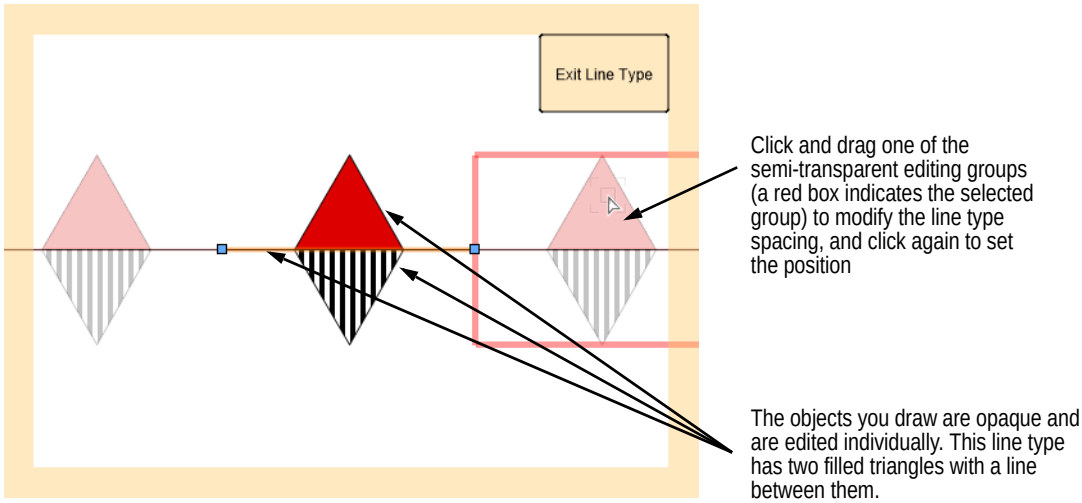
   The New Line Type dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies a name for the line type resource; this name is displayed in the Resource Browser and Attributes palette
Simple	Select this option to create a custom dashed line; you can either drag levers on the graph, or enter a specific line length and gap length for each dash section
Dash graph	Levers indicate the distance (in page inches or page millimeters) between dash sections; a section consists of a line and a gap. To add a dash section, drag a new lever from the right edge of the dash graph; to delete a section, drag a lever from the rightmost section all the way to the right. A dash line type can have up to five sections.
<< or >>	Selects a highlighted section for editing; alternatively, click a section to highlight it
Line Length	Specifies the length of the line, in page units, for the highlighted dash section
Gap Length	Specifies the length of the gap, in page units, for the highlighted dash section
Scale with line thickness	Maintains the proportion between the line type and the line thickness
Complex	Select this option to create a line type with 2D graphic elements
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

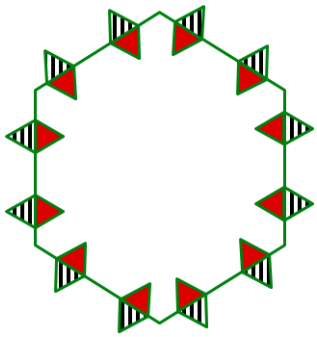
Parameter	Description
Page Based/World Based Units	Specifies the units for the line type when it is used in a drawing: • **Page** sets the line type size according to the page dimensions. When applied, the line type automatically scales relative to the page environment. This is useful for annotation objects that should always remain the same size on the "page," regardless of the layer scale. • **World** sets the line type size according to world dimensions; its size varies depending on the layer scale.

4. Click **OK** to save the settings.
   - For a simple line type, the definition is complete, and the new line type displays in the Resource Browser.
   - For a complex line type, a line type editing window opens. A red baseline across the middle of the window indicates the path of the line type, but it is not part of the line type geometry.
5. To create the complex line type, draw the line type geometry with the desired 2D objects, including fills and text (symbols and parametric objects are not allowed). All line type definitions have a black pen color and .03 mm line thickness. When you create an object, a partially transparent repetition of the object displays to the left and right of it. You can add multiple objects to the pattern.
6. To adjust the spacing of a complex line type, click one of the repetitions and drag it as needed; click again to set the new position. To hide the repetitions during editing, Ctrl-click (Mac) or right-click (Windows) on an empty area of the editing window, and deselect the **Display Repetitions** option on the context menu.



7. When you are done drawing the complex line type geometry, click **Exit Line Type**. The line type resource is saved with the specified name.

   Line types are saved in Vectorworks drawing files. If the file is not saved and the line type is not default content, the line type is lost when you exit the program.



Sample line type applied to an object, with 1 mm line thickness and dark green pen color

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Line Types

Apply line types to specific objects using either the Attributes palette or the Resource Browser.

Attributes can also be applied by class settings (see "Setting Class Attributes" on page 181) or transferred with the **Eyedropper** tool.

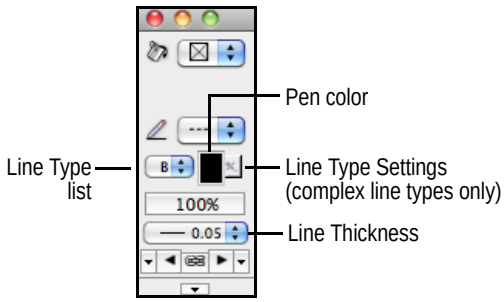
Applying a Line Type from the Attributes Palette

To apply a line type from the Attributes palette and specify the line type settings:

1. Select the object(s), and then select **Line Type** from the Attributes palette pen style list.

   If no line types are defined, and default content is not enabled in Vectorworks preferences, you are prompted to add a default line type definition.

2. To change to a different line type, click on the line type list and select a different type from the list of default content or the current file's content (see "Resource Libraries" on page 219).



3. Adjust the pen color and line thickness for the selected object(s) as needed.
4. If a complex line type is selected, customize it by clicking the **Line Type Settings** button next to the **Line Type** list. The Line Type Local Mapping Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Line type selector	Select a line type resource from either the default content or the current file's content
Use Local Mapping	Select to enable entry in the mapping fields, to customize the line type for this object; deselect to return to the default mapping
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Segment Length	Specifies the distance for a single instance of the line type (in the file's current units). Objects in the line type will scale symmetrically when this is changed; for example, if the segment length is halved, each segment becomes half as long and half as tall.
Flip Horizontally/ Flip Vertically	Select to flip the line type orientation horizontally, vertically, or both

5. If desired, click **Preview** to view the results of the line type settings.
6. Click **OK** to apply the line type settings.

Applying a Line Type from the Resource Browser

To apply a line type from the Resource Browser:

1. Select the object(s).
2. From the Resource Browser, select the resource to be applied. Right-click (Windows) or Ctrl-click (Mac) and select **Apply** from the context menu. (Alternatively, double-click the resource to apply it to the selection or drag the resource onto an object.)
3. Adjust the pen color and line thickness for the selected object(s) as needed.
4. If a complex line type is selected, customize the line type settings from the Attributes palette as described in "Applying a Line Type from the Attributes Palette" on page 1130.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Transferring Attributes
Defining Line Types
Editing Line Type Definitions

Editing Line Type Definitions

Changes to a line type definition affect all instances of that line type in the drawing file.

To edit a line type definition:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.
2. Select the desired line type. Right-click (Windows) or Ctrl-click (Mac), and select **Edit** from the context menu.
3. The Edit Line Type dialog box opens; select whether to edit the Geometry or Settings.
   - If you edit the settings, the Edit Line Type Settings dialog box opens next; select whether to use page-based or world-based units. Click **OK** to return to the drawing area.
   - If you edit the geometry, the line type editing window opens next to allow editing of the line components. Adjust the line type definition as described in "Defining Line Types" on page 1128.
4. Any instance of the edited line type in the drawing file changes according to the new definition.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

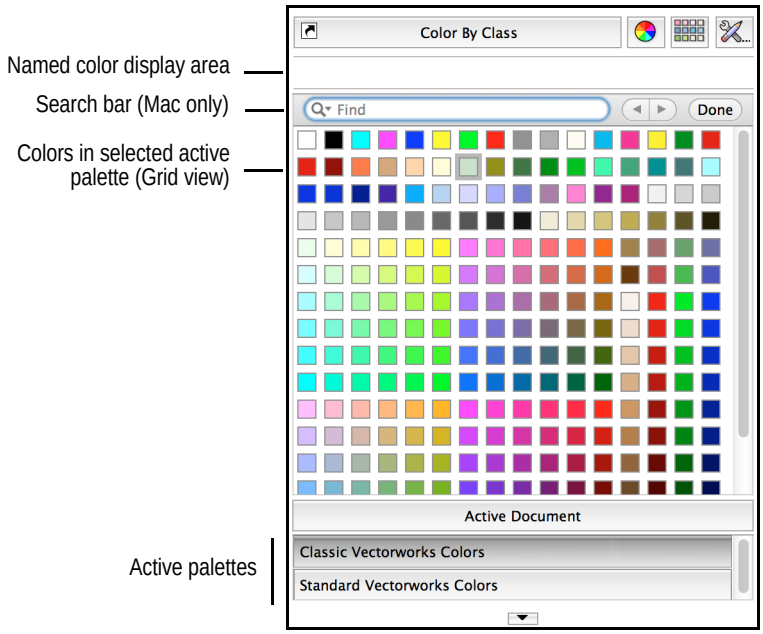
Defining Line Types
Applying Line Types

Applying Colors

The Color Palette set provides colors for selection and specifies which colors are available in a Vectorworks file. To access the Color Palette set from the Attributes palette, click on one of the Fill color boxes (select either a Solid or Pattern fill style) or on one of the Pen color boxes (select any Pen style).

A similar interface is available from other areas in the application where colors are specified, although the Color By Class option is only available when accessed from the Attributes palette.

See “Setting Default Colors and Palettes” on page 1133 for information on how to set up and activate color palettes.



Select an active color palette, and then select the color to apply.

Select **Color By Class** to use the color attributes set by the object’s class.

- If **Use at Creation** is enabled for the object’s class, the object’s color is automatically set by the class.
- If **Use at Creation** is disabled for the class, the object only takes on the class color when the **Color By Class** option is selected.

To find a named color on Windows, begin typing; the letters display in the named color display area and the closest color match is selected. Press the Tab key to cycle through the closest matches; pause for several seconds to restart the search.

To find a named color on the Mac, type in the search bar under the named color display area. Search options are available from the Search bar menu.

- Ignore Case: when selected, search function will not be case sensitive
- Wrap Around: When selected, search function will search entire palette selection rather than starting from the currently selected color
- Contains: When selected, search function will search a color’s entire name for the entered text
- Starts With: When selected, search function will only show colors that begin with the entered text
- Full Word: When selected, search function will ignore partially matched words

After typing the name of the desired color, click the left and right arrows to cycle through the list of matching colors. Click **Done** to close the search bar, press Command-F to open the search bar again.

See “Setting Class Attributes” on page 181.

Setting Default Colors and Palettes

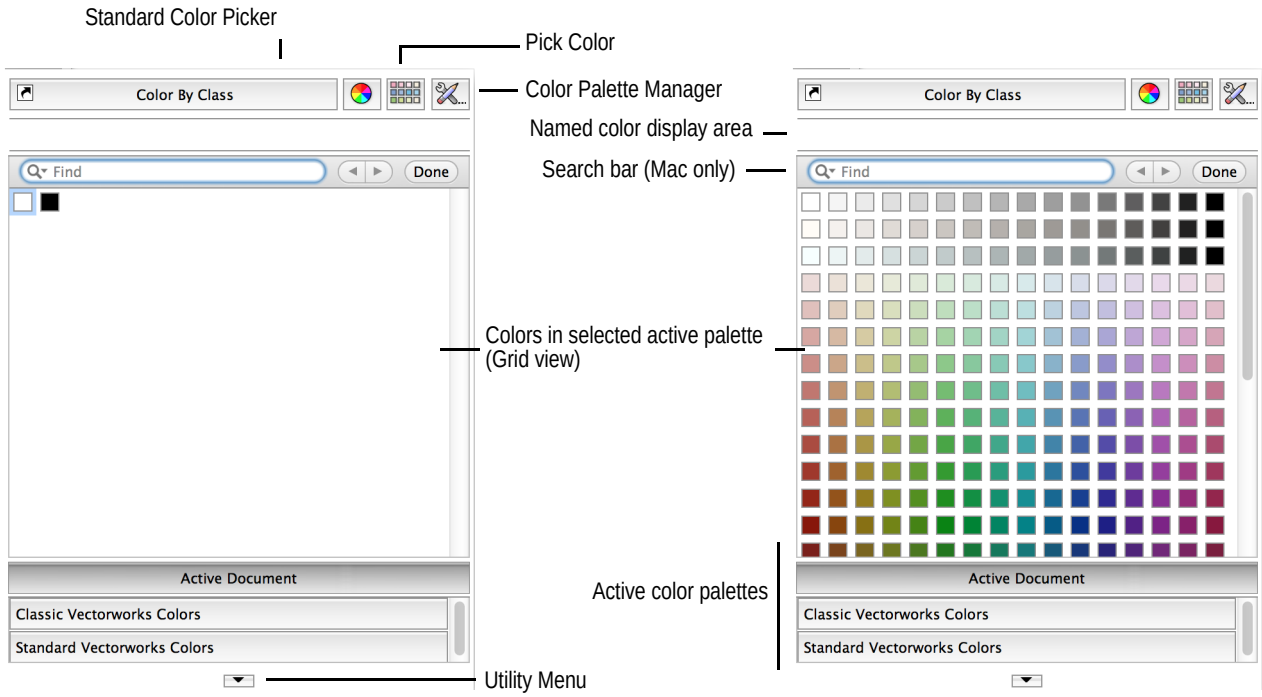
The Color Palette set opens when the Fill color or Pen color box is clicked from the Attributes palette. The Color Palette set provides colors for selection and specifies which colors are available in a Vectorworks file. Any set of unlimited colors can be included; colors are organized in standard and/or custom color palettes.

In a new file, the active document color palette is the only palette displayed in the Color Palette set, and it contains only the colors black and white. As colors are added to the file from color palettes as well as resources, the active document color palette expands to display all the colors used in the file.

To display and manage the colors available for the file:

1. From the Attributes palette, click on one of the color boxes for Fill color (either a Solid or Pattern fill style must be selected) or Pen color (any pen style can be selected). A similar interface is available from other areas in the application where colors are specified, although the Color By Class option is only available when accessed from the Attributes palette.

   The Color Palette set opens, displaying the colors in the active color palette.



Color Palette set in a new file, showing the default active color palettes

Color Palette set in a file with active color palettes

Click to show/hide the parameters.

Parameter	Description
Color by Class	When using the Color Palette set to specify an object's color, causes the object to take on the color attribute set by the object's class; see "Applying Colors" on page 1132
Standard Color Picker	Opens the operating system's color picker for selecting individual colors; see "Selecting Standard Operating System Colors" on page 1135

Parameter	Description
Pick Color	Opens the Pick Color dialog box, for selecting a color from available color palettes; see “Selecting a Color from a Color Palette” on page 1135
Color Palette Manager	Opens the Color Palette Manager dialog box, for managing available color palettes and activating additional palettes; see “Managing Color Palettes” on page 1136
Named color display area	Displays any color name information associated with the selected color. This area also serves to find a named color in the active color palette. To easily find a named color, begin typing; the letters display in the named color display area and the closest color match is selected. Press the Tab key to cycle through the closest matches; pause for several seconds to start the search over.
Search Bar (Mac only)	To find a named color on the Mac, type in the search bar under the named color display area. Search options are available from the Search bar menu. ● Ignore Case: when selected, search function will not be case sensitive ● Wrap Around: When selected, search function will search entire palette selection rather than starting from the currently selected color ● Contains: When selected, search function will search a color’s entire name for the entered text ● Starts With: When selected, search function will only show colors that begin with the entered text ● Full Word: When selected, search function will ignore partially matched words After typing the name of the desired color, click the left and right arrows to cycle through the list of matching colors. Click Done to close the search bar, press Command-F to open the search bar again.
Colors in active palette	Displays the selected active palette’s available colors in either Grid or List view; these colors are available for use in the file
Active palettes	Lists palettes which have been made active for the file with the Color Palette Manager; click on a palette name to display its colors for use
Utility Menu	Opens a utility menu to control the color palette sorting and display

2. Either select a color from one of the file’s color palettes, or click off of the Color Palette set (in the drawing window, for example) to close the color window and set the file’s available colors.

Color Palette Utility Menu

The options selected from the color palette Utility Menu button at the bottom of the main Color Palette set indicate the color sorting and display.

To set the display and sorting of colors and palettes:

1. From the Color Palette set, click the **Utility Menu** button to open the **Utility** menu.

Menu/Command	Action
View Color Palette as	Select to view the colors in the Color Palette set as a grid of colored squares, or a list of colors and associated names. (In either view, color names are always shown in the display area above the colors as the mouse moves over the colors.)

Menu/Command	Action
Sort Color Palette by	Specify the sorting method for the active color palette; select Color to sort by hue (HSV values), Manual to sort according to the order set in the Color Palette Manager, or Alphabetical to sort in ascending or descending order by color name
Place Color Palettes at	Select whether to display active color palettes at the top or bottom of the Color Palette set
Shrink Grid to Fit	In Grid view, color lists of less than 16 rows can be displayed with a Color Palette set option that shrinks to fit the available colors

2. Select a utility command to change the color palette display or sorting option.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selecting Standard Operating System Colors
Selecting a Color from a Color Palette
Managing Color Palettes
Creating or Editing Custom Color Palettes
Applying Colors
Setting Default Colors and Palettes
Creating a Color Chart
The Attributes Palette

Selecting Standard Operating System Colors

When specifying or creating color palettes is not necessary, the operating system's color picker adds individual colors to the active document color palette.

 To select a standard operating system color:

1. From the Color Palette set, click the **Standard Color Picker** button.

   The Colors (Mac) or Color (Windows) dialog box opens.
2. Select a color from among the standard operating system colors, and click **OK**.
3. The color is added to the active document color palette, and becomes available for use in the file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Default Colors and Palettes
Selecting a Color from a Color Palette
Managing Color Palettes
Creating or Editing Custom Color Palettes
Applying Colors
Creating a Color Chart

Selecting a Color from a Color Palette

If a file requires colors beyond those available for the standard system color picker, but you do not wish to add and manage palettes in the Color Palette set, select a color from a specific color palette.

 To add colors from a color palette:

1. From the Color Palette set, click the **Pick Color** button.

The Pick Color dialog box opens, listing the color palettes available. The Colors in Active Document palette is always available. Other palettes in the list either ship with the Vectorworks software, or have been added with the Color Palette Manager. The Vectorworks Fundamentals product ships with Classic Vectorworks Colors and Standard Vectorworks Colors palettes; many other palettes are available with Vectorworks Design Series products.

Click to show/hide the parameters.

Parameter	Description
Color Palette list	Lists the color palettes from which colors can be selected. Click on the header to sort the palettes in ascending/descending alphabetical order. Select a palette to display its colors on the right.
Color List	Lists the colors in the selected palette; click on the header to sort by color or by color name. Select a color to add it to the Colors in the Active Document palette.
Color Grid	Displays the colors in the selected palette as a grid; select a color to add it to the Colors in Active Document palette
Color Information	Displays the color information of the selected color, including Name; Red, Green, Blue (RGB); Cyan, Magenta, Yellow, and Black (CMYK); and Hue, Saturation, and Value (HSV). This information also displays on a screen tip when you position the cursor over a color.

2. Select a color palette from the left, and then select the color from the list or grid of colors on the right.
3. Click **OK** to add the color to the active document palette, and make it available for use in the file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Default Colors and Palettes
Selecting Standard Operating System Colors
Managing Color Palettes
Creating or Editing Custom Color Palettes
Applying Colors
Creating a Color Chart

Managing Color Palettes

The Color Palette Manager determines which color palettes are available in the main Color Palette set and the Pick Color dialog box. With the Color Palette Manager, create or import custom color palettes, activate palettes for use in the file, add and delete colors from custom color palettes, and purge unused colors.

 To manage the file's color palettes:

1. From the Color Palette set, click the **Color Palette Manager** button.

   The Color Palette Manager dialog box opens. The palettes available here are located in the Color Palettes library of the default content; see "Resource Libraries" on page 219. The Vectorworks Fundamentals product ships with Classic Vectorworks Colors and Standard Vectorworks Colors palettes; many other palettes are available with Vectorworks Design Series products.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Color palette list	Lists the file's available color palettes. Click on a header to sort the palettes by active status or color palette name. Click in the **Active** column of a selected palette to activate the palette and add it to the available palettes in the main Color Palette set.
Color List	Lists the colors in the selected palette; click on the header to sort by color or by color name
Color Grid	Displays the colors in the selected palette as a grid
New	Opens the New Palette dialog box
Edit	Opens the Edit Palette dialog box
Duplicate	Copies the selected color palette and appends an incrementing number to the copy name (change or rename the copied palette by clicking **Edit**)
Delete	Deletes the currently selected color palette; only custom palettes can be deleted, but not the Colors in Active Document or standard color palettes
Purge Unused	When the Colors in Active Document palette is selected, removes any colors that are not in use in the file from the Colors in Active Document palette. Referenced colors remain in the active document color palette, even from an object that has been removed. Purging allows accumulated colors to be trimmed.

2. When the list of color palettes and active color palettes is set, click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Default Colors and Palettes
Selecting Standard Operating System Colors
Selecting a Color from a Color Palette
Creating or Editing Custom Color Palettes
Applying Colors
Creating a Color Chart

Creating or Editing Custom Color Palettes

Custom color palettes can be created or edited from the Color Palette Manager.

Type the first few letters of a palette (or color, when the cursor is in the color list) to quickly select a color.

To create or edit a custom color palette:

1. Open the Color Palette Manager as described in "Managing Color Palettes" on page 1136.
2. Click **New** to create a new palette, or **Edit** to make changes to the currently selected palette.

   The New Palette or Edit Palette dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies a name for a new color palette, or edits the name of a custom color palette
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Color List	Lists the colors in the custom palette; click on the header to sort by number in the list, color, or color name. To change the order of the colors, click in the # column and drag the selected color up or down in the list. (Select **Manual** from the color palette **Utility** menu to sort the colors in the main Color Palette set in this order; see "Color Palette Utility Menu" on page 1134)
Color Grid	Displays the colors in the custom palette as a grid
New	Opens the New Color dialog box, for adding a new color from the operating system's color picker
Edit	Opens the Edit Color dialog box, for editing the selected color using the operating system's color picker
Delete	Deletes the currently selected color(s) from the custom color palette
Get	Opens the Pick Color dialog box, for adding one or more colors selected from the available color palettes
Lighten	Incrementally lightens the selected color
Darken	Incrementally darkens the selected color
RGB Blend	Blends two selected colors according to their Red, Green, Blue (RGB) values, to create the specified number of new colors (up to 1000)
HSV Blend	Blends two selected colors according to their Hue, Saturation, and Lightness (HSV) values, to create the specified number of new colors (up to 1000)
Import	Imports all the colors from the Colors in Active Document palette of another current version Vectorworks file

3. When the colors have been edited or added to the custom color palette, click **OK**.

   The custom color palette is listed in the Color Palette Manager. By default, custom color palettes are saved in the user folder (see "User Folders Preferences" on page 57).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Default Colors and Palettes
Selecting Standard Operating System Colors
Managing Color Palettes
Selecting a Color from a Color Palette
Applying Colors
Creating a Color Chart

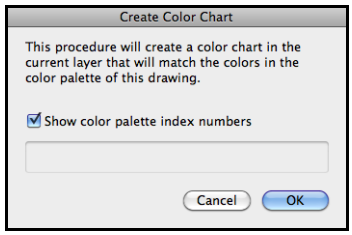
Creating a Color Chart

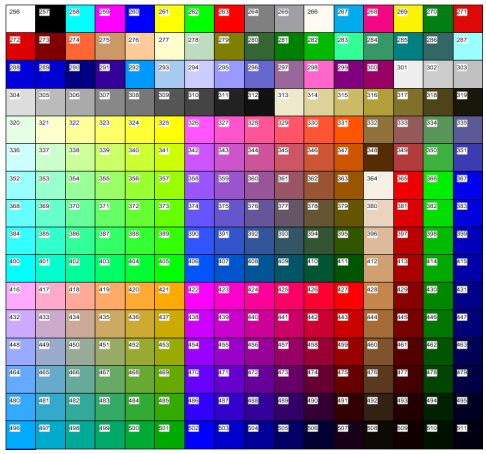
This command creates a color chart in the active layer of the current file that reflects the colors in the color palette of the file. It can be used as a print color guide.

To create a color chart:

1. Select **Tools > Utilities > Create Color Chart**.
2. Confirm that a color chart of the active layer in the current file should be generated.

   If desired, select **Show color palette index numbers** to show the color index numbers in the chart. The color chart should be viewed in Top/Plan.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~





The index numbers are internal to the color palette and always remain in the same order.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

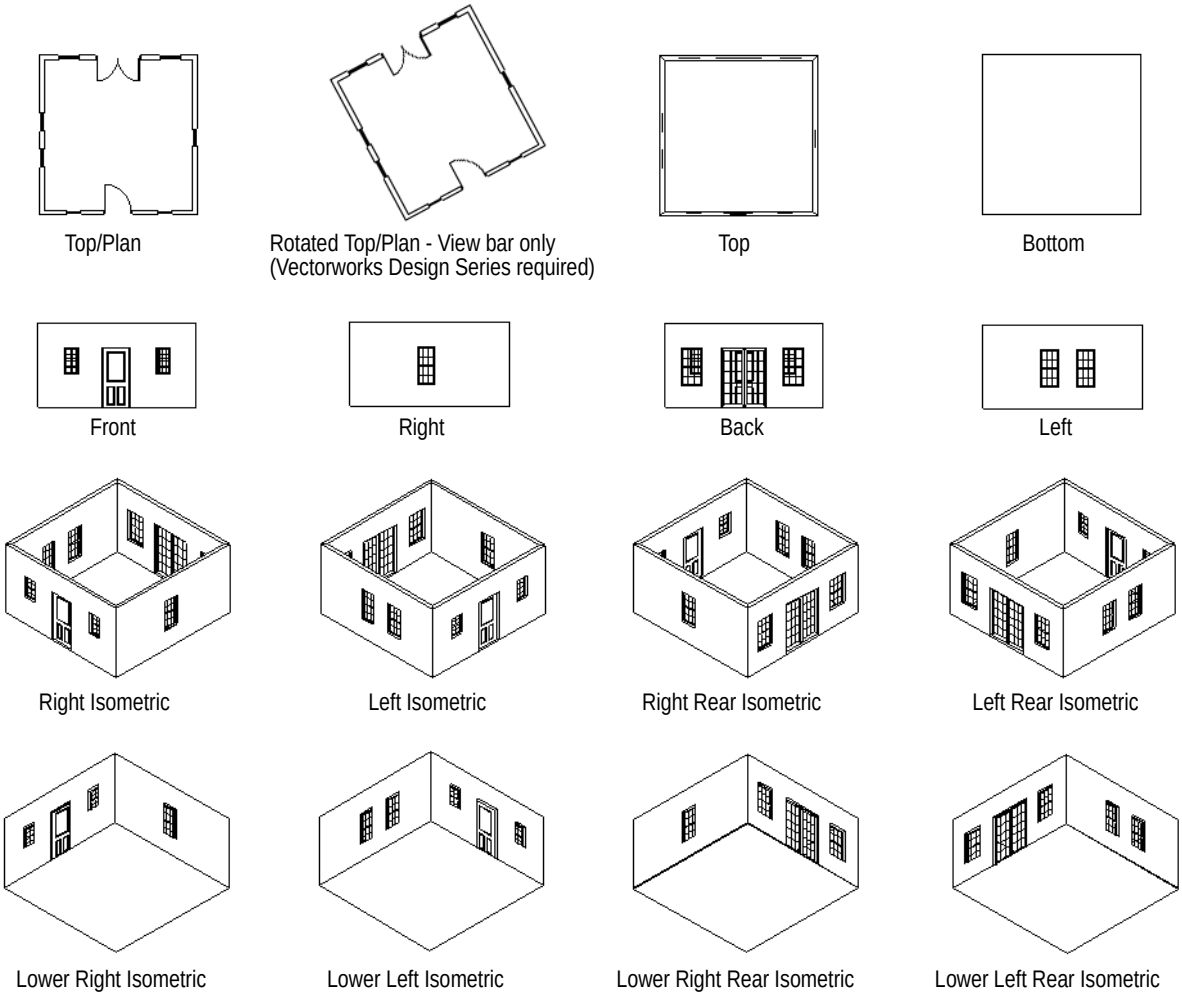
Setting Default Colors and Palettes
Selecting Standard Operating System Colors
Managing Color Palettes
Selecting a Color from a Color Palette
Creating or Editing Custom Color Palettes
Applying Colors
Transferring Attributes
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewing the Drawing

Using Standard Views

The items under **View > Standard Views** change the representation and the orientation of the drawing area. The view options can be divided into four basic categories:

Category	Description
2D View	Select the **Top/Plan** command to view the drawing in 2D; use this for annotations, title blocks, and 2D illustrations. **Rotated Top/Plan** view (Vectorworks Design Series required) is only available from the View bar when a rotated top/plan view exists.
3D Elevations	Use the **Top**, **Front, Right**, **Bottom, Back**, and **Left** commands to view 3D elevations
3D Representations Above the Active Layer Plane	Use the **Right Isometric**, **Left Isometric**, **Right Rear Isometric,** and **Left Rear Isometric** commands to view 3D representations above the active layer plane
3D Representations Below the Active Layer Plane	Use the **Lower Right Isometric**, **Lower Left Isometric, Lower Right Rear Isometric**, and **Lower Left Rear Isometric** commands to view 3D representations below the active layer plane



To change among the different views:

Select **View > Standard Views**, and then select a view.

The view changes and a check mark displays in front of the active view in the **Current View/ Standard Views** list.

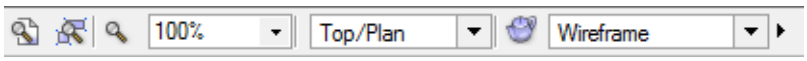
The reference for the view, whether working plane or active layer plane, may depend on the selection in the Working Planes palette (see "Working Plane View and Modes" on page 1178).

To switch views rapidly, use shortcuts on the numeric keypad. These shortcut keys also work to change the view of a selected viewport on a sheet layer.

Keypad Number	View
0	Top/Plan Rotated Top/Plan
1	Left Isometric
2	Front
3	Right Isometric
4	Left
5	Top
6	Right
7	Left Rear Isometric
8	Rear
9	Right Rear Isometric

When a rotated top/plan view exists (Vectorworks Design Series required), the keypad number 0 invokes the Rotated Top/Plan view instead of standard, unrotated Top/Plan view. Press the 0 key a second time to toggle to the standard unrotated Top/Plan view.

The View bar also provides quick access to the standard view commands. Click the **Current View** and select a standard view from the list that displays. If the view is something other than a standard view, "Custom View" displays as the current view.



Current View/Standard Views list

Vectorworks Nomad

If you have a Vectorworks Service Select subscription, you can use the Vectorworks Nomad app to view and present your 3D drawings rendered in OpenGL on your iOS mobile device. Click here for more information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Vectorworks Cloud Services

Projection

The projection commands alter the way the program displays the 3D geometry of the drawing on a 2D screen.

The perspective-related projection commands add distortion to the drawing so that objects that are farther away appear smaller than objects that are closer. These views closely approximate how the 3D model would display in the real world. The perspective view can be uncropped or cropped (see "Cropped and Uncropped Perspective Views" on page 1144).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

In addition to the standard 2D Plan projection, there are nine 3D projection modes.

To switch projection modes:

Select **View > Projection**, and then select a projection.

Projection Mode	Description
2D Plan	Matches the projection for a normal 2D drawing; use this view to draw 2D objects
Orthogonal	Displays an undistorted 3D projection of the drawing—objects display at their exact size regardless of their distance from the active layer plane
Narrow Perspective	Displays a projection of the drawing similar to a telephoto lens
Normal Perspective	Displays a projection of the drawing similar to a portrait lens
Wide Perspective	Displays a projection of the drawing similar to a wide angle or fisheye lens
Set Custom Perspective	Select to set a custom perspective numerically. The Set Perspective Distance dialog box opens. Type the custom perspective value in the **Distance** field, and then click **OK**. The lower the value, the wider the angle of perspective.
Oblique Cavalier (30° or 45°)	The Cavalier modes show an undistorted front view along with depth (also known as full-depth axonometric projections). The lines along the Z axis (which show the depth of an object) are represented in true length. This distorts the overall image of the object and makes it appear deeper. These modes make precise measurements possible, either from a printout of the drawing or from the screen image.
Oblique Cabinet (30° or 45°)	The Oblique Cabinet modes (also known as half-depth axonometric projections) are similar to Oblique Cavalier. However, the depth lines are shortened by 50%. This distorts the actual length of these lines, but represents a more natural view of the object. These modes can still be used to take measurements of vectors perpendicular to the projection plan (along the Z axis) from a printed drawing, but the results must be multiplied by two to obtain the actual dimension.

OpenGL and the Renderworks product do not support the Oblique projections; files with an OpenGL or Renderworks render mode are converted from an Oblique projection to Orthogonal projection when saved. To render Oblique projections, use one of the Polygon render modes, or use Hidden Line mode.

A Vectorworks preference sets the preferred projection, applied whenever the drawing is switched from Top/Plan into a 3D view; see “3D Preferences” on page 54.

In an uncropped perspective view, scrolling within the drawing, as well as using the **Pan** and **Zoom** tools, will result in perspective camera movement while the view remains centered in the window. When there are no objects in the current view to use as a reference during movement, a special indicator displays to help orient the movement.

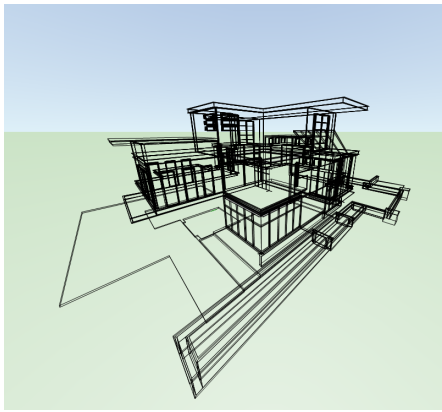
The empty view indicator provides visual 3D navigation feedback when there are no objects in the uncropped perspective view

The **Translate View** tool can also be used to move through a model and interactively adjust the perspective distortion while in a perspective projection; see “Translate View” on page 1149.

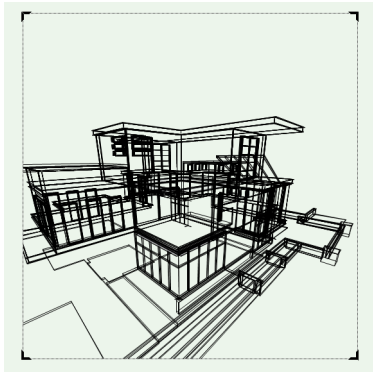
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cropped and Uncropped Perspective Views
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cropped and Uncropped Perspective Views



Uncropped perspective projection



Cropped perspective projection

Perspective projection can be in an uncropped modeling view or in a cropped presentation view. The cropped perspective projection is set for the entire document on the Display tab of the Document Preferences dialog box; see “Document Display Preferences” on page 60.

In a cropped view, a clipping frame appears around the perspective view window, which is useful for presenting, for example, a model elevation in perspective view. Adjust the clipping frame by dragging the corners to increase or decrease the size of the window. In older Vectorworks version drawings, this was the only method available for displaying perspective views.

In an uncropped view, a shaded horizon representation is displayed in the background. The horizon provides a visual indication of the vertical position of the observer with respect to objects in the scene. The horizon does not print or export. The green color represents the ground below the horizon, and the blue color represents the sky above it, helping with orientation. The color of the ground and sky can be changed in the Interactive Appearance Settings dialog box; see “Configuring Interactive Display” on page 116.

Simulating Movement

Several tools simulate movement over and through the drawing.

When you use these tools, a rendered model may display with OpenGL rendering temporarily, even when set to render with a different render mode. When the movement stops, the program renders the new view of the model. Additionally, for a highly complex drawing, the program may temporarily remove some of the detail to speed up the movement. When the movement stops, the detail returns.

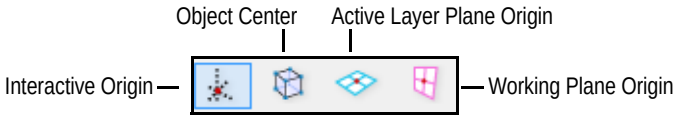
These tools have certain Tool bar buttons that are true modes, and others that act as command buttons. When some of these tools are in use, the Data bar provides display-only information to help orient the view.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Flyover

The **Flyover** tool simulates movement over and around a real-world model.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

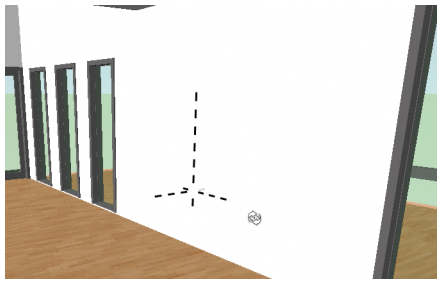
To control the movements of the **Flyover** tool, drag the mouse around a selected center of rotation.
The Tool bar buttons set the center of rotation.



Mode	Description
Interactive Origin	Sets the center of rotation with a mouse click
Object Center	Sets the center of rotation to the center of the selected objects; if no objects are selected, sets the center of rotation to the center of the visible objects
Active Layer Plane Origin	Sets the center of rotation to the center of the active layer plane
Working Plane Origin	Sets the center of rotation to the origin of the current working plane

To fly over a drawing:

1. Click the **Flyover** tool from the Basic palette, or from the 3D Modeling tool set or Visualization tool set.
2. Select the mode from the Tool bar.
3. In Interactive Origin mode, click to specify the center of rotation for the flyover movement.
4. The center of rotation is indicated by dashed axis lines.



To fly over the drawing, click on the drawing and drag in the direction of movement while you hold the mouse button. To stop the movement, release the mouse button. Alternatively, use shortcuts on the keyboard to fly over the drawing.

Mouse Movement	Shortcut Keys	Description
Move left or right		Turns left or right about the selected center of rotation
Move up or down		Moves up or down about the selected center of rotation
Move in toward center	Alt + move right (Win) or Option + move right (Mac)	Moves the view toward the center of rotation (in Perspective projection)
Move outward from center	Alt + move left (Win) or Option + move left (Mac)	Moves the view outward from the center of rotation (in perspective projection)
Move down toward active layer plane	Alt + move up (Win) or Option + move up (Mac)	Moves down toward the active layer plane (in perspective or orthogonal projection)

Mouse Movement	Shortcut Keys	Description
Move up from active layer plane	Alt + move down (Win) or Option + move down (Mac)	Moves up from the active layer plane (in perspective or orthogonal projection)
	Shift key	Constrains the rotation to the global Z axis, or to the Z' axis of the active working plane (when **Working Plane Mode** is enabled on the View bar)
	Ctrl key (Win) or Command key (Mac)	Draws selected objects in full detail, and hides deselected objects (so that the redraws are faster)

Activating the Flyover Tool Temporarily

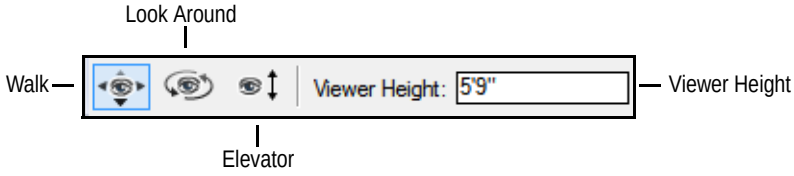
While another tool is active, press and hold the mouse wheel button and the Ctrl key (Windows) or Control key (Mac) simultaneously to activate the **Flyover** tool. Orient the view as desired, and release the mouse. The previous tool becomes active again automatically.

This feature will not work properly if the wheel button is assigned a custom function in the mouse setup. For example, if the wheel button is set to perform a delete when clicked, a wheel click in the Vectorworks program deletes rather than activates the flyover tool. (The specific setting required for this feature depends on the type of mouse being used.)

Walkthrough

The **Walkthrough** tool simulates movement through a 3D model.

The tool operates in perspective projection. Three modes, the **Viewer Height** parameter, and various modifier keys provide flexibility to manipulate the viewer position and view angle before and during the walkthrough.



Mode	Description
Walk	Moves the viewer through the drawing
Look Around	Changes the view angle up, down, right, and left while the viewer remains stationary
Elevator	Raises and lowers the viewer height

Mode	Description
Viewer Height	Specify the viewer height above the ground plane. This value updates automatically after Elevator mode is used; it also reflects the viewer height as set by any other viewing tool.

When the **Walkthrough** tool is in use, the Data bar displays the following information to help orient the view.

Data Bar	Description
Yaw	Displays the angle (0 – 360 degrees) between the walking direction and the positive Y axis, to describe the direction of movement in the active layer plane. A yaw of 90 degrees indicates movement in the direction of the positive X axis. A yaw of 180 degrees indicates movement in the direction of the negative Y axis.
Pitch	Displays the viewing angle. A value of 0 degrees indicates that the viewer is looking straight ahead; a value of 45 degrees indicates that the viewer is looking up at an angle 45 degrees from horizontal.
Viewer X/Y/Z	Displays the position of the viewer relative to the active layer plane

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Walking Through a Model
Looking Around a Model
Setting Viewer Height

Walking Through a Model

To conduct a walkthrough of a model:

1. Click the **Walkthrough** tool from the Visualization tool set and click **Walk** from the Tool bar.

   The projection automatically switches to perspective.

2. Click on the drawing and hold the mouse button to perform the walkthrough.

   The first mouse click inside the drawing, indicated by a small cross, establishes the control point of the walkthrough. The control point lets you manage the pace of the walkthrough. The farther away from the control point the mouse moves, the faster the walkthrough. Return the mouse to the control point to slow or pause the movement. To stop the walkthrough, release the mouse button.

3. Move the mouse up (forward), down (backward), left, and right to walk through the drawing.

   The Data bar provides real-time information about the walkthrough.

   To change the viewing angle or viewer height as desired during the walkthrough, combine the movement with a modifier key.

Modifier Key	Description
Alt key (Win) or Option key (Mac)	Move the mouse to change the look angle up/down (pitch) or right/left (yaw), following the mouse movement; the viewer's position and height remain fixed. Move the mouse at an angle to change both the pitch and yaw at the same time and create a sweeping view.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifier Key	Description
Shift key	Move the mouse to change the viewer height; the viewer's X,Y position and viewing angle remain fixed

Walkthrough
Looking Around a Model
Setting Viewer Height

Looking Around a Model

 To look around a model from a fixed viewer point:

1. Click the **Walkthrough** tool from the Visualization tool set and click **Look Around** from the Tool bar.

   The projection automatically switches to perspective.
2. Click on the drawing and hold the mouse button to look around.

   The first mouse click inside the drawing, indicated by a small cross, establishes the control point of the look around. The control point lets you manage the pace of the look around. The farther away from the control point the mouse moves, the faster the movement. Return the mouse to the control point to slow or pause the movement. To stop the look around, release the mouse button.
3. Move the mouse in the direction you want to look.

   The Data bar provides real-time information while looking around.

   To change the viewer height while looking around, if desired, press the Shift key and move the mouse up or down.

Walkthrough
Walking Through a Model
Setting Viewer Height

Setting Viewer Height

 To change the viewer height:

1. Click the **Walkthrough** tool from the Visualization tool set and click **Elevator** from the Tool bar.

   The projection automatically switches to perspective.
2. Click on the drawing and hold the mouse button while you move the mouse up or down. The Data bar provides real-time information about the viewer height.

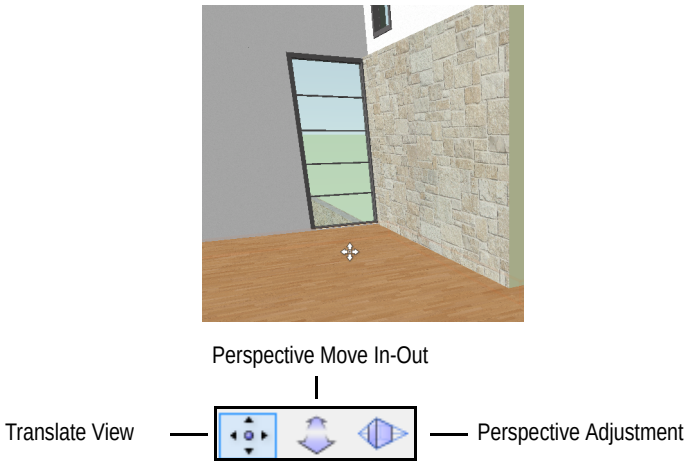
   The first mouse click inside the drawing, indicated by a small cross, establishes the control point of the movement. The control point lets you manage the pace of the movement. The farther away from the control point the mouse moves, the faster the movement. Return the mouse to the control point to slow or pause the movement. To stop the operation, release the mouse button.

   Alternatively, if you know the desired viewer height, set the value in **Viewer Height** on the Tool bar. This field can be edited in any mode.

Walkthrough
Walking Through a Model
Looking Around a Model

Translate View

The **Translate View** tool changes the view of a 3D model, especially when in a perspective projection.



Mode	Description
Translate View	Moves the drawing along the screen X axis when the cursor moves left or right, and along the screen Y axis when the cursor moves up or down
Perspective Move In-Out	In perspective projection, drag the mouse up and down to move the view forward (closer) or backward (farther away) along the current sightline
Perspective Adjustment	In perspective projection, adjusts the distortion by moving the mouse up (more distorted) or down (less distorted); similar to creating a custom perspective view by selecting **View > Projection > Set Custom Perspective**

Double-click the **Translate View** tool to refresh the drawing view.

To translate the view:

1. Click the **Translate View** tool from the Visualization tool set.
2. Depending on the selected mode, click on the drawing and drag while you hold the mouse button to translate the drawing from side to side or up and down, move in and out of the drawing, or adjust the perspective. To stop the movement, release the mouse button.

   Press the Ctrl key (Windows) or Command key (Mac) to draw selected objects in full detail, and hide deselected objects (so that the redraws are faster).

   When the **Translate View** tool is in use, the Data bar displays the following information to help orient the view.

Data Bar	Description
Viewer X/Y/Z	Displays the position of the viewer relative to the active layer plane
L/R	Displays the movement to the left and right of an imaginary axis perpendicular to the screen
U/D	Displays the movement up and down from an imaginary axis perpendicular to the screens
I/O	Displays the movement in and out of the drawing along an imaginary axis perpendicular to the screen
Perspective distance	In Perspective Adjustment mode, displays the perspective distance

Cropped and Uncropped Perspective Views

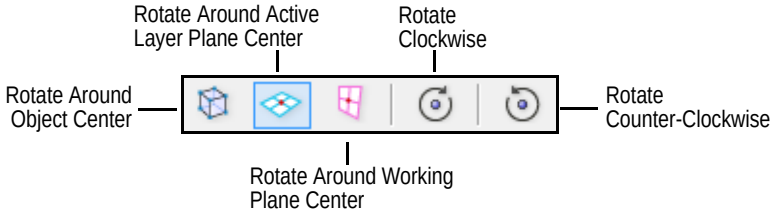
Rotating the 3D View

To rotate the 3D view, either use the **Rotate View** tool and the mouse, or specify rotation values with the **Rotate 3D View** command.

Rotating with the Mouse

The **Rotate View** tool rotates the view clockwise or counter-clockwise in 3D.

To control the movements of the **Rotate View** tool, drag the mouse or click one of the mode buttons. The first three Tool bar buttons set the center of rotation. The last two modes perform precise, incremental rotation movements in the direction specified.



 To rotate the 3D view:

1. Click the **Rotate View** tool from the Visualization tool set.
2. Select the mode from the Tool bar.

Mode	Description
Rotate Around Object Center	Uses the center of selected objects as the rotation center
Rotate Around Active Layer Plane Center	Uses the center of the active layer plane as the rotation center
Rotate Around Working Plane Center	Uses the origin of the current working plane as the rotation center

3. To rotate the view, click on the drawing and drag while holding the mouse button. Move the cursor to rotate the view in the direction of the mouse movement. Release the mouse to stop the rotation. Alternatively, use the two Tool bar buttons.

Mode	Mouse Movement/Key	Description
	Move left or right	Rotates about the screen Y axis (or the screen Z axis if the mouse is at the perimeter of the drawing)
	Move up or down	Rotates about the screen Y axis (or the screen Z axis if the mouse is at the perimeter of the drawing)
Rotate Clockwise		Each click rotates the view in a clockwise direction about the selected center of rotation
Rotate Counter-Clockwise		Each click rotates the view in a counter-clockwise direction about the selected center of rotation
	Ctrl key (Windows) or Command key (Mac)	Draws selected objects in full detail, and hides deselected objects (so that the redraws are faster)

When the **Rotate View** tool is in use, the Data bar displays the following information to help orient the view.

Data Bar	Description
Azimuth	Displays the active layer plane angle (in degrees) between the positive X axis and the direction from the selected center of rotation to the viewer
Elevation	Displays the angle (in degrees) between the selected center of rotation and the plane which is parallel to the active layer plane and passes through the view origin
Roll	Displays (in degrees) the amount of rotation about the selected rotation center

Rotating Precisely

The **Rotate 3D View** command provides a precise way to rotate the 3D view. Enter rotation values directly, or rotate the view in increments. A preview displays the rotation movements before the actual view is rotated.

To rotate the 3D view:

1. Select **View > Rotate 3D View**.

   The Rotate 3D View dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Parameters are absolute	Starts the view rotation from a top (absolute) view
Parameters are relative	Starts the view rotation from the current view
Interactive Rotation	
+, - X/Y/Z buttons	The plus and minus buttons for each axis rotate the view by the increment specified
Angle Increment	Specifies the rotation increment (in degrees) for the interactive rotation buttons
Rotation by Value	
Rotation on X/Y/Z	Specifies the rotation value (in degrees, minutes, and seconds) for the X, Y, and/or Z axes

2. Specify the rotation criteria, and then click **OK** to rotate the 3D view as specified.

Using a SpaceNavigator Mouse

The Vectorworks program supports both the SpaceNavigator and SpaceExplorer editions of the 3Dconnexion SpaceNavigator™ mouse. The SpaceNavigator driver must be installed; see www.3dconnexion.com/support/

SpaceNavigator movement affects the current view; when you cycle through movement modes with the SpaceNavigator's left button, the Vectorworks program's Message bar displays a notice.

The effects of the SpaceNavigator mouse on the view depend on the current view and projection, as well as the selected mode.

For best results while navigating, render the drawing with OpenGL rendering.

Cycle through the SpaceNavigator modes for various types of navigation.

SpaceNavigator Mode	Description	Vectorworks View or Projection
Using Walkthrough Navigation	Moves forward, back, left, right, up, and down; looks left, right, up, and down	Perspective

SpaceNavigator Mode	Description	Vectorworks View or Projection
Using Unconstrained Navigation	Moves forward, back, left, right, up, and down; looks left, right, up, and down, and rolls left and right. This mode is the most flexible, but can be challenging to control; decreasing axis sensitivity from the SpaceNavigator control panel is recommended (press the right Navigator button)	Perspective
Using Flyover Navigation	Functions like the **Flyover** tool, with spinning and tilting motions. If objects are selected, the fixed point of rotation is the objects' center; otherwise, the fixed point of rotation is the internal origin	Perspective, Isometric
Using 2D Navigation	Allows pan and zoom	Top/Plan, Perspective, Isometric

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Flyover
Walkthrough

Unified Layer View

The **Unified View** command accesses a modeling mode to view, snap to, select, and edit objects in multiple design layers within a unified 3D coordinate system. This mode is active by default to see the entire model and edit multiple objects across layers easily, or to align objects across layers, without having to create a viewport or use layer links.

In a unified view, the Vectorworks program aligns all visible design layers in the drawing file with the active layer, and displays them using the active layer's scale, lighting options, render mode, and Renderworks background (Renderworks required). The unified view also provides a unified coordinate system; the Z coordinates of all objects are expressed relative to the active layer plane. Other layers are above or below the active layer, depending on their elevation's relationship to the active layer.

The active layer's layer options setting (**View > Layer Options**) determines whether other layers can be rendered and snapped to, and whether objects on other layers can be selected and modified. For rendering, select **Show Others** or **Gray Others**. For both rendering and snapping, select **Show/Snap Others** or **Gray/Snap Others**. When **Show/Snap/Modify Others** is selected, objects on other layers can be selected and they can be modified.

Click here for a video tip about this topic (Internet access required).

Setting Unified View Options

Set the unified view options to obtain the desired behavior.

To set unified view options:

1. Select **View > Unified View Options**. Alternatively, double-click the **Unified View** button on the View bar.

   The Unified View Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Display Screen Objects	Displays screen plane objects from all visible objects in the unified view, and allows access to 2D tools
Only on Active Layer	Only displays screen plane objects for the active layer
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Restore Original Views when exiting Unified View mode	Returns layers to their original view status and Renderworks background when turning off unified view. When deselected, the layer views remain aligned when turning off unified view, and the current layer's Renderworks background applies to all layers.
Ignore Layers with Different Scale	Does not include layers with a scale that is different from the active layer in the view; select this option to hide annotation or detail layers
Show only Layers belonging to Stories	Displays only layers that belong to stories in the unified view; select this option to exclude layers, such as detail layers, from the view
Center View on Activated Layer	When selected, centers the view on the active layer in unified view; the view does not shift when switching to another layer

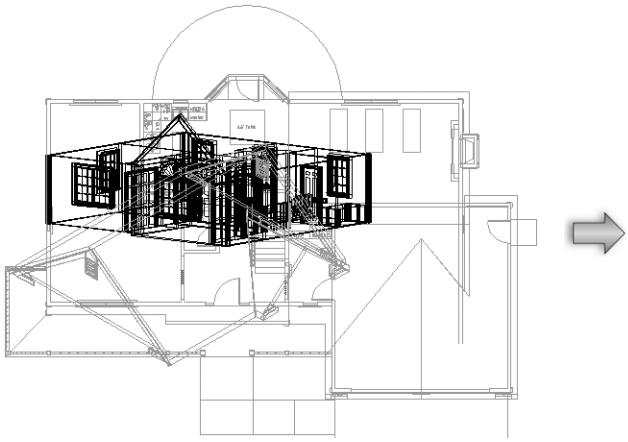
2. Set the parameters and click **OK**.

Unified View

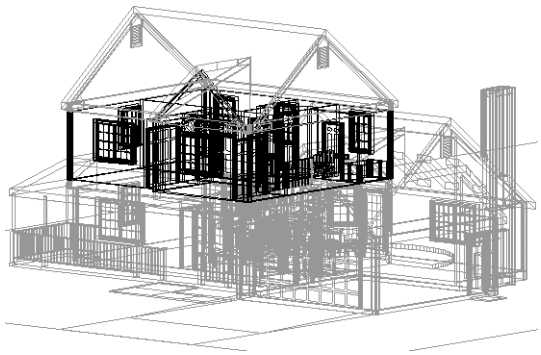
To enable unified view:

1. Select **View > Unified View**. Alternatively, click the **Unified View** button on the View bar.
2. To return to a normal layer view, select **View > Unified View** again.

   A check mark next to the **Unified View** command indicates that the feature is activated.

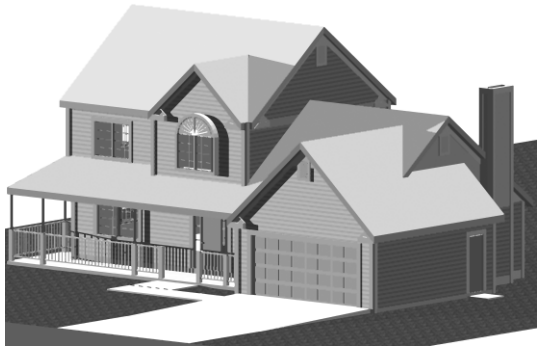


In normal layer view, the active layer is in a rotated view; other inactive layers are in Top and Left Isometric views



The Unified View command aligns all visible layers to the active layer's rotated view

With the layer options set to Gray Others and OpenGL rendering, a transparent effect can be created for the grayed layers



With the layer options set to Show Others, all visible layers can be stacked and rendered

With the layer options set to Show/Snap/Modify Others, objects on any layer can be selected and modified

Setting a 3D View

The **Set 3D View** command sets a precise 3D view at a specified viewing angle, height, and perspective. Normally, the command is executed from 2D Top/Plan view; however, the command can also set a 3D view from any of the 3D views.

To set a 3D view:

1. Select **View > Set 3D View**.
2. Click and draw a line to indicate the view direction; the line starting point indicates the viewer position and the line end point indicates the point the viewer is looking towards (look toward point).

   The Set 3D View dialog box opens.
3. Specify the 3D view criteria.

Click to show/hide the parameters.

Parameter	Description
Viewer Height	Specifies the height of the viewer at the start point of the drawn line
Look Toward Height	Specifies the height of the view destination at the end point of the drawn line
Perspective	Select the desired perspective, or use the current setting to keep the perspective unchanged

4. Click **OK**. The 3D view is adjusted as specified.

Setting a Camera View in Renderworks

Viewing a Model with the Clip Cube

The **Clip Cube** command allows you to temporarily clip away portions of a 3D model to see and work inside of the model. Only objects inside the cube are visible and snappable. Less clutter makes it easier to locate snap points or to view a specific region of interest in a large 3D model.

The clip cube feature only works in Wireframe and OpenGL rendering modes. Select objects in the area to view, and then select the **Clip Cube** command to create a cube that bounds those objects. Once the cube is created, use the **Selection** tool to push or pull the cube faces to adjust the size. Use the cube's editing frame to rotate it or drag it to another location.

To use the clip cube feature:

1. In a 3D view, select the objects to be visible inside the cube. (If no objects are selected, the cube will bound all currently visible objects.)
2. Set the rendering mode to Wireframe or OpenGL.
3. Select **View > Clip Cube**.

   Alternatively, add the **Clip Cube** shortcut button to the Tool bar; see "Setting Quick Preferences" on page 64.

   The view is clipped to show only the area of the model that contains the selected objects; the objects are surrounded by a transparent cube. By default, cross-section areas of solid objects are displayed in red, along the plane where the clip cube cuts.
4. To modify the clip cube, click an edge on the cube with the **Selection** tool. An editing frame with X, Y, and Z axes displays on its bottom face. During editing, the view changes to show the objects within the cube at any given time.

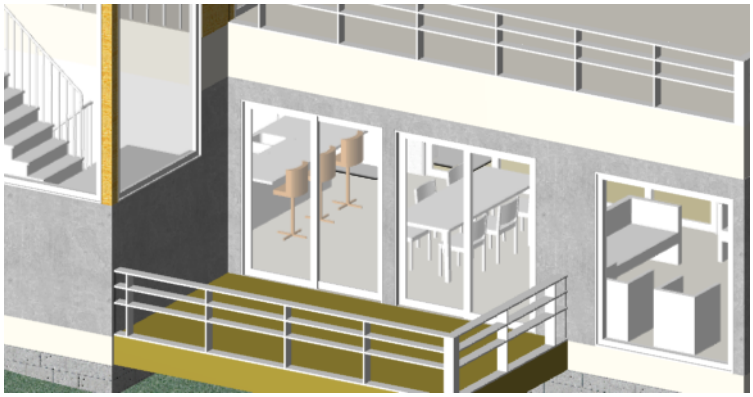
Action	Description
To adjust the cube dimensions	Similar to the **Push/Pull** tool, a face of the cube becomes highlighted when the cursor is over it. Click a highlighted face and drag it as needed; click again to set to the new location.
To rotate the cube	Click one of the four rotation handles on the edges of the editing frame on the bottom face of the clip cube. Drag to rotate the cube, and then click to set the new location.
To move the cube	Click the move handle in the center of the editing frame. Drag to move the cube, and then click to set the new location.

5. Edit the objects within the clip cube as needed. Keep in mind the following:
   - If you edit a symbol, the clip cube will be reset to the symbol boundary; when you exit the symbol editing mode, the cube returns to its original boundary.
   - While the clip cube feature is enabled, you can create a different cube on each design layer. However, if the unified view feature is enabled, only one clip cube can be created.
6. To save the current clip cube view for later use, select **View > Save View**, and select the option to **Save View Orientation** (see "Creating Saved Views" on page 189).
7. Select the **Clip Cube** command again to disable the clip cube and return to a full model view on all design layers.

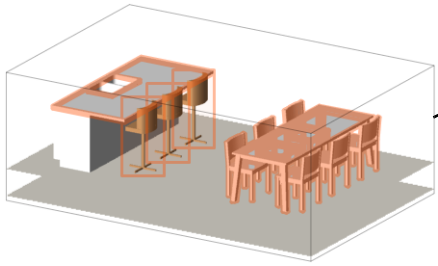
   (A check mark next to the **Clip Cube** command indicates that the feature is activated.)

   If you hold down the Ctrl key (Windows) or Command key (Mac) the next time you select the **Clip Cube** command, the last clip cube you had on this design layer is restored.

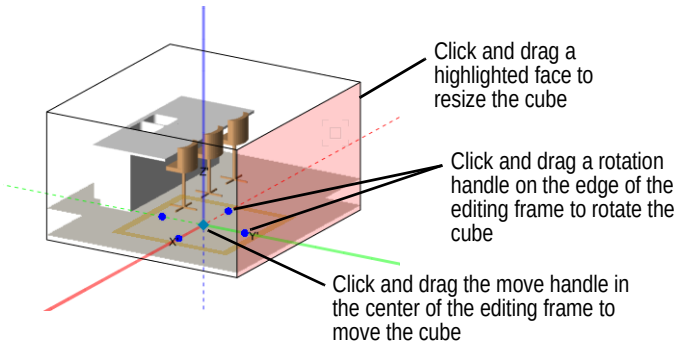
A 3D model with OpenGL rendering and unified view enabled; objects within the building are obscured



Select objects of interest, and then select View > Clip Cube to create a transparent cube around them; objects outside the cube become temporarily invisible



In editing mode, you can resize, rotate, and move the cube; the objects that are visible change according to the cube location



D Creating a Section Viewport from a Clip Cube

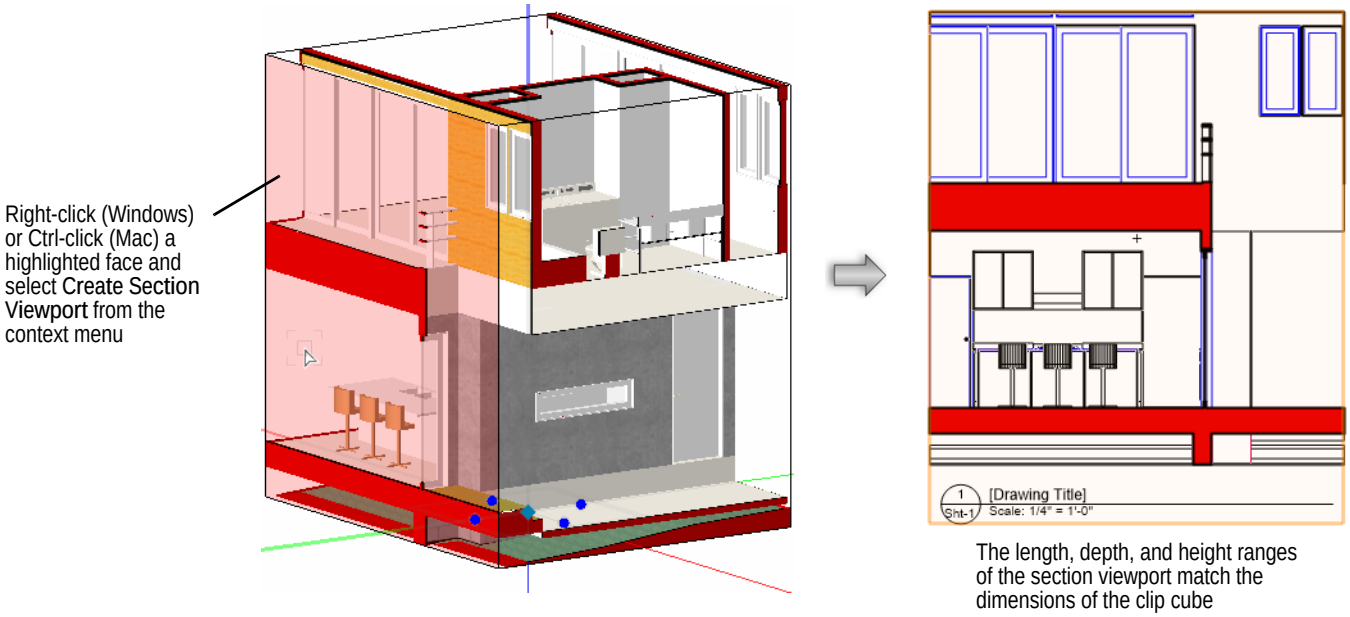
After you create a clip cube, select the **Selection** tool, and then right-click (Windows) or Ctrl-click (Mac) a highlighted vertical face of the clip cube.

Select **Create Section Viewport** from the context menu. The Create Section Viewport dialog box opens.

Create the viewport either on a sheet layer or design layer, as follows:

- “Creating a Section Viewport on a Sheet Layer” on page 1625
- “Creating a Section Viewport on a Design Layer” on page 1628

By default, the cross-section areas are shown with red fill. These areas are in the Section Style class, if you want to change the fill color.



Click here for a video tip about this topic (Internet access required).

X-ray Select Mode

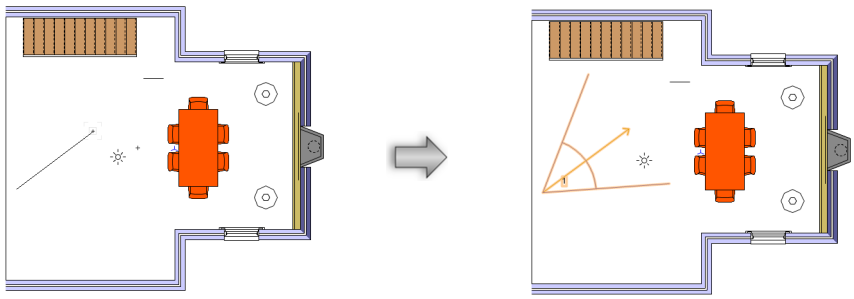
Setting a Camera View in Renderworks

The Renderworks product contains a method of setting a camera view with the **Renderworks Camera** tool. Specific attributes, such as camera focal length, field of view, height, and aspect ratio can be set.

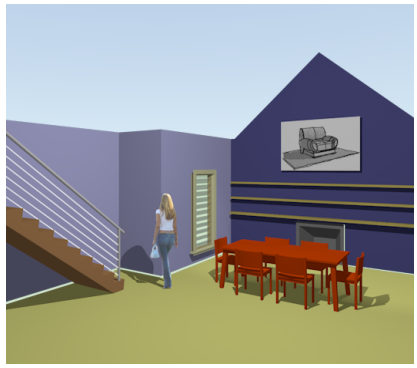
To insert a Renderworks camera:

1. Select the **Renderworks Camera** tool from the Visualization tool set.
2. Click in the design layer to specify the camera location. Click again to indicate the camera look-to point.

   If this is the first time a camera object has been inserted in this session, the Renderworks Camera Object Properties dialog box opens. Click **OK**.



3. Click **Display Camera View** in the Object Info palette, or simply double-click on the camera, to switch to the 3D camera view. To return to Top/Plan view, click **Top/Plan View** in the Object Info palette.



4. The camera properties can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Camera Height	Sets the camera height; at placement, the camera is set to a default height of 1500 mm or 5’0”. If a Z value is also specified for the camera object, the total camera height is the sum of the Z height and the **Camera Height**.
Look To Height	Sets the height of the camera look-to point; at placement, the look-to height is set to a default height of 1500 mm or 5’0”. If a Z value is also specified for the camera object, the total look-to height is the sum of the Z height and the **Look To Height**.
Top/Plan View	Switches to a Top/Plan view of the camera
Display Camera View	Switches to display the 3D camera view (double-clicking on the camera in a 2D view also switches to camera view)
Fine Tune Camera View	Opens the Perspective View Controls dialog box, for making fine adjustments to the camera view controls
Projection	Select Perspective or Orthogonal projection for the camera view; Perspective creates a cropped or uncropped perspective view of the model from the camera view, while Orthogonal is useful for an elevation view (for example, a skewed elevation view of a building which is not in a standard view). Available parameters depend on the selected projection.
Render Mode	Selects a render mode for the 3D camera view
Aspect Ratio	Sets the aspect ratio of the perspective clipping window; the clipping window can also be set to the page size or to a custom aspect ratio
Custom Aspect	When a custom **Aspect Ratio** is selected, enter the custom ratio
For Film Size of	Specifies the camera film size, and determines the focal length of the camera (has no effect on the camera view)
Focal Length is	Displays the camera focal length, based on film size
Field of View	Specifies the view angle; set to a default angle of 65 degrees. Use the control point to set the view angle on the 2D camera with the mouse.
For DPI of	Calculates the pixel size when exporting the camera view (has no effect on the camera view)
Pixel Size is	Displays the pixel size based on the DPI setting

Parameter	Description
Crop Frame Scale %	Scales the size of the clipping window frame, when in a cropped perspective projection, and also scales the perspective distance. As a result, both the crop frame and the drawing appear scaled.
Left/Right Tilt Angle	Tilts the camera to the left or right, for more accurate perspective matching
Camera Name	Specifies a name for the camera, which can be displayed or hidden in 2D view; move the camera name text control point to adjust the position of the name
Camera Display	Select a camera display mode; the camera name only displays in 2D • None: Hides the camera in 2D and 3D views, and hides the camera name • 2D: Displays the camera in 2D view, but hides the camera name, and hides the camera in a 3D view • 3D: Displays the camera's perspective clipping window bounding box in 2D view, displays the camera in 3D view, and hides the camera name • 2D + 3D: Shows the camera in 2D and 3D views, but hides the camera name • 2D + Name: Shows the camera and camera name in 2D view, but hides the camera in 3D views • 3D + Name: Shows the camera in 3D view, shows the camera name in 2D view, but hides the camera in 2D view (displays a locus instead) • 2D + 3D + Name: Shows the camera and camera name in 2D view, and shows the camera in 3D views The 3D camera view displays the bounding box of the camera view, the view line, and the look-to end point. Displaying a camera in a 3D view allows it to be easily selected during design development, and it can be hidden later for final presentation
Auto Update 3D View	When selected, automatically updates the 3D camera view with every parameter change; for complex models, deselect when making several parameter changes, and then either re-select **Auto Update 3D View** or click **Display Camera View** to update the camera view with any parameter changes
Auto Center 3D View	Automatically centers the view when **Display Camera View** is clicked, or the camera is double-clicked

Adjusting the Camera View
Linking the Camera View to a Sheet Layer Viewport
Setting a 3D View
Managing Lights and Cameras with the Visualization Palette
Projection

Adjusting the Camera View

The camera view settings can be fine tuned in real-time, and the display attributes specified.

To adjust the camera view:

1. Click **Fine Tune Camera View** from the Object Info palette of a selected camera object.

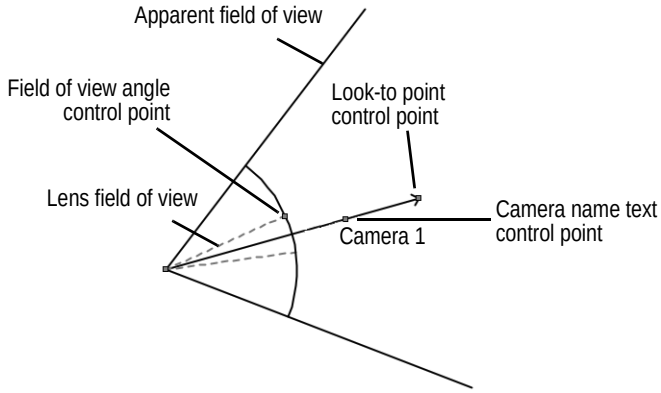
The Perspective View Controls dialog box opens. Either use the sliders or enter values to adjust the camera settings; changes are reflected automatically in the drawing. To interactively view changes as you move the sliders, render in wireframe or OpenGL render mode.

For **Camera Height**, **Look To Height**, and **Camera Distance**, the slider range is proportional to the displayed value. To expand the slider range, enter a larger value.

Click to show/hide the parameters.

Parameter	Description
Camera Height	Moves the camera vertically
Look To Height	Sets the height of the look-to point, which effectively tilts the camera vertically
Camera Pan	Pans the camera, rotating it about its axis within a +/- 20 degree range, as if it were on a tripod
Camera Move Left/Right	Rotates the camera about the look-to point, within a +/- 20 degree range
Camera Distance	Moves the camera towards or away from the look-to point
Focal Length (zoom)	Acts like a zoom lens; sets the lens focal length, from 10 to 200 mm and changes the field of view angle
Perspective	Increases or decreases the perspective effect, making the perspective lines vanish more or less steeply about the look-to point; works most effectively when the look-to point is at the center of the scene or object being viewed
Clip Frame Aspect Ratio	Sets the aspect of the perspective crop window; use in conjunction with the **Clip Frame Size** to obtain the desired window size
Clip Frame Size	Sets the scale of the perspective crop window; use in conjunction with the **Clip Frame Aspect Ratio** to obtain the desired window size
Render Mode	Selects a render mode for the 3D camera view

2. A camera object can be copied to insert additional camera views. In addition, the 2D camera display attributes can be set by fill and pen foreground and background colors in the Attributes palette. Control points adjust the field of view angle, the look-to point, and the camera name location.



Fill/Pen Color Attribute	Parameter
Fill foreground	Sets the field of view angle color when the camera height is higher than the look-to height
Fill background	Sets the text color for the camera name

Fill/Pen Color Attribute	Parameter
Pen foreground	Sets the color of the view line connecting the camera to the look-to point, as well as the color of the 3D camera representation
Pen background	Sets the color of the angle of view lines

The divergence between the apparent field of view angle and the lens field of view angle increases as the difference between the camera and look-to heights increases. When the camera and look-to heights are equal, the lens field of view angle line (dashed gray line by default) is not visible.

Setting a Camera View in Renderworks
Linking the Camera View to a Sheet Layer Viewport
Setting a 3D View
Applying Colors
Managing Lights and Cameras with the Visualization Palette

Linking the Camera View to a Sheet Layer Viewport

The view of a sheet layer viewport can be controlled by a Renderworks camera. Establish the view with the Renderworks camera on the design layer, and then create a sheet layer viewport linked to the camera. The camera view can be edited from the viewport.

Renderworks cameras cannot be linked to section viewports.

To link a Renderworks camera to a sheet layer viewport:

1. Create and adjust the Renderworks camera as described in “Setting a Camera View in Renderworks” on page 1157. A camera linked to a viewport can be in perspective or orthogonal projection.
2. Select the Renderworks camera, and then select **View > Create Viewport**.
3. An alert dialog box asks whether the camera should be used for the viewport’s view. Select **Yes** (click **Always do the selected action** to always use a selected Renderworks camera for the view when creating viewports).
4. The Create Viewport dialog box opens. Enter a viewport name and drawing title, and select the sheet layer to place it on. The view and projection parameters are set by the camera, and therefore appear dimmed; the camera’s render mode does not affect the viewport’s render mode. Change any other parameters as needed (see “Creating a Sheet Layer Viewport from a Design Layer” on page 1616).
5. Click **OK**.

   The viewport is created on the specified sheet layer, with its view, projection, and perspective distance set to that of the Renderworks camera. In the Object Info palette, the **RW Camera** status changes to Yes.

   The camera becomes part of the viewport and can be edited (or deleted), changing the viewport’s view parameters, by editing the camera through the viewport. An existing sheet layer viewport can also be linked to a Renderworks camera. See “Editing a Linked Renderworks Camera” on page 1654.

   Click here for a video tip about this topic (Internet access required).

Setting a Camera View in Renderworks
Adjusting the Camera View
Editing a Linked Renderworks Camera
Creating a Sheet Layer Viewport from a Design Layer
Editing a Design Layer Displayed in a Viewport

Fit to Objects

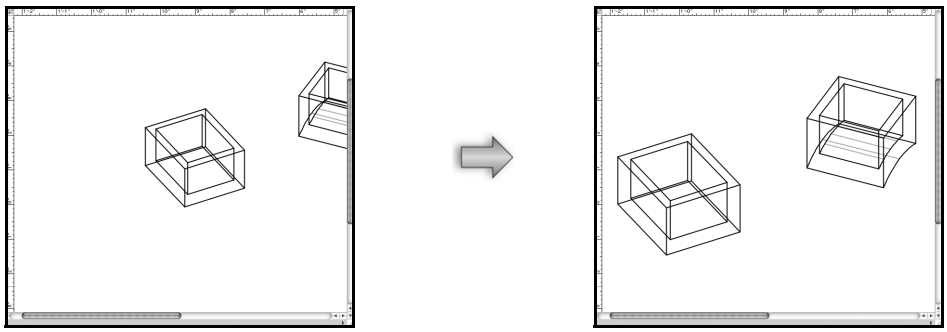
The **Fit to Objects** command provides an easy way to zoom in and out of a drawing. There are two options: fit the window around all the objects in the drawing, or fit the window around a particular object or set of objects.

When in an uncropped perspective projection, the view direction is preserved, and the viewpoint is adjusted to fit the model or object within the drawing window.

To fit the drawing window around all visible objects in the drawing:

1. Ensure that the current layer contains the object(s) to view, and that no objects are selected.
2. Select **View > Zoom > Fit to Objects**. Alternatively, click the **Fit to Objects** button on the View bar.

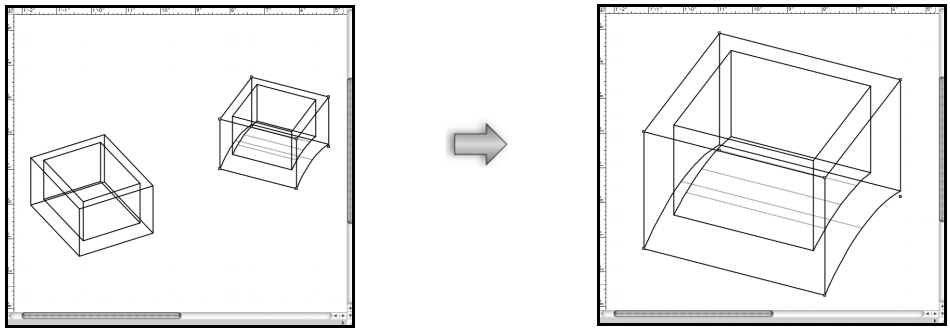
   The program zooms in or out so that all the objects in the drawing display in the drawing area.



To fit the view to a particular object or set of objects:

1. Ensure that the current layer contains the object(s) to view.
2. Select the object or objects to view.
3. Select **View > Zoom > Fit to Objects**. Alternatively, click the **Fit to Objects** button on the View bar.

   The program zooms in to display only the selected object(s) in the drawing area.



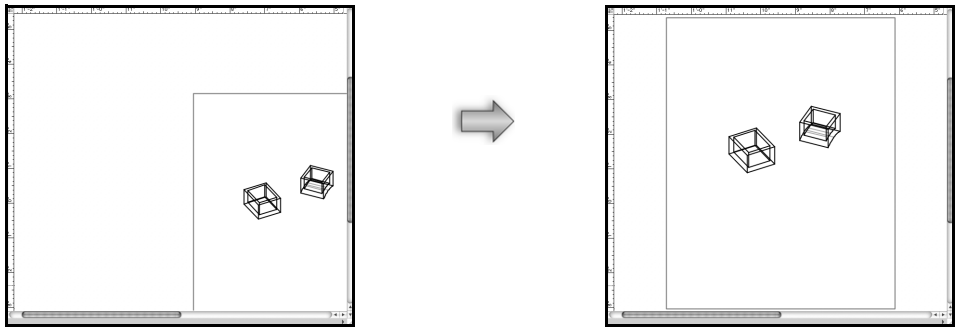
Fit to Page Area

The **Fit to Page Area** command displays the entire print area in the drawing window. For a single page drawing, this command displays the entire page. For a drawing with two or more pages, all of the pages display at one time.

To fit the print area in the window:

Select **View > Zoom > Fit to Page Area**. Alternatively, click the **Fit to Page Area** button on the View bar.

The program zooms in or out to display all pages in the print area.



Viewing History

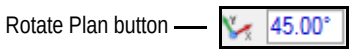
The program automatically records a history of the last 50 changes in drawing view, including projection, perspective and zoom changes. Click the **Previous View** and **Next View** buttons on the View bar to move through the change history stack.

On a Windows system with a five-button mouse, buttons 4 and 5 on the mouse (which invoke the Back and Forward commands in web browsers) invoke Previous View and Next View, respectively.

D Rotating the Plan

In a design layer, the view may need to be rotated to match a drawing angle. When the plan is rotated, the view switches to Rotated Top/Plan, and all existing layers are rotated together.

The View bar has a button for quick access to the **Rotate Plan** command.



To rotate the plan:

1. Select **View > Rotate Plan**, or click **Rotate Plan** from the View bar.
2. Click to indicate the center of plan rotation, and then click again to indicate the angle of plan rotation. The angle snaps to existing objects.

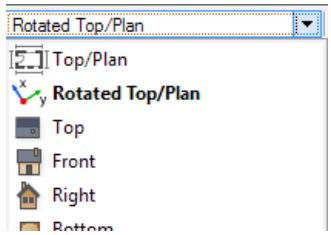
   The temporary reference line indicates the line that will become horizontal after the operation. The temporary arrow within the arc indicates the direction of the rotation. The rotation is referenced to a horizontal line, unlike the **Rotate** tool.

   The angle of plan rotation can also be specified from the floating data bar, or entered directly in the View bar. The drawing rotates around plan center (0, 0) when the angle is specified in the View bar.
3. All layers in the drawing rotate to match the specified angle. (If some layers were in a 3D view, they will lose this view since all layers in the drawing are set to Rotated Top/Plan; confirm that this is the desired action.) Symbols placed and objects drawn while in this view match the current rotation angle.

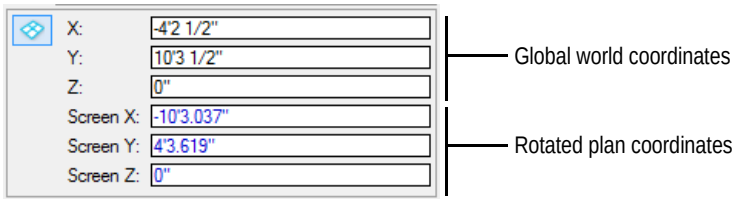
   At the bottom left of the drawing window, the position of the X - Y axis indicator shows the drawing rotation angle and direction.

To disable the rotation animation, see “Interactive Preferences” on page 56.

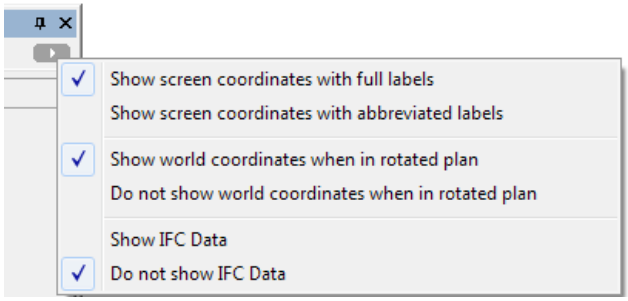
The View bar indicates that the drawing has been rotated; the icon next to the layers in the Layers list changes, and the **Current View** list displays Rotated Top/Plan. If you switch to another view and want to return to the rotated top/plan view, select Rotated Top/Plan from the **Current View** list on the View bar.



In addition, in a Rotated Top/Plan view, the rulers display in a blue color, and the Object Info palette for objects in rotated plan display both global world coordinates and rotated plan coordinates (depending on the Object Info palette preference).



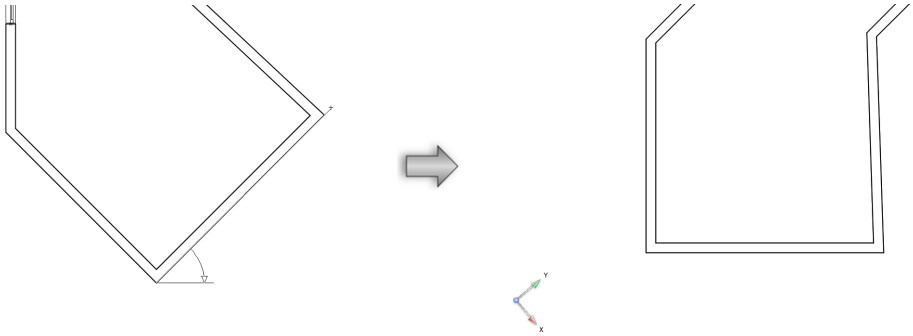
The display of the Object Info palette coordinates in rotated plan view is controlled by a preference, accessed from the **Coordinate/IFC** menu at the top right corner of the Object Info palette:



Click to show/hide the parameters.

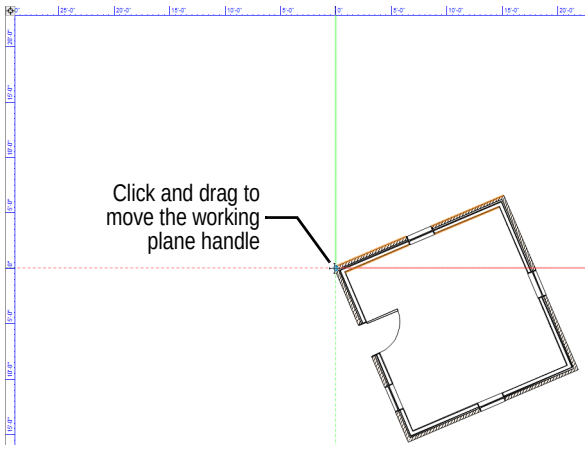
Parameter	Description
Show screen coordinates with full labels	Displays screen coordinates (rotated plan coordinates) with Screen X, Screen Y, and Screen Z labels in the Object Info palette
Show screen coordinates with abbreviated labels	Displays screen coordinates with abbreviated X’, Y’, and Z’ labels
Show world coordinates when in rotated plan	Displays world coordinates in addition to screen coordinates when in rotated plan view
Do not show world coordinates when in rotated plan	Hides world coordinates when in rotated plan view
IFC options (Vectorworks Architect or Landmark required)	For information on IFC data, see “Assigning IFC Data to Objects” on page 1746

Once the drawing has been rotated, a different rotation angle can be specified with **Rotate Plan** in the View bar.



The green reference line indicates the unrotated, horizontal direction. If the **Rotate Plan** command is selected again, match this reference line to un-rotate the view, returning to the world coordinate system. Alternatively, to un-rotate the view, select **View > Standard Views > Top/Plan**, select **Top/Plan** from the View bar, or enter an angle of 0 for **Rotate Plan** in the View bar.

In rotated plan view, a working plane is automatically present, with its origin at the rotated plan pivot point and its X' axis horizontal to the screen. Click on the X' or Y' axis to drag the working plane to a new location. The X' and Y' coordinates adjust from this point and the local coordinates are reflected on the rulers.



An elevation view can be easily set based on the working plane position. Click **Working Plane Views** from the View bar, and then select a standard view such as Front. To return to the rotated 2D view, select Rotated Top/Plan from the **Current View** list on the View bar.

The plan rotation can be saved as a view and restored later by selecting the view for display. Select **Save View Orientation** in the Save View dialog box to save the plan rotation (see "Creating Saved Views" on page 189).

Saved views with rotation information display with a rotated icon in the View column of the Organization dialog box. Sorting by the View column separates non-rotated views from the rotated views.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The View Bar
Creating Saved Views

Using Working Planes

The working plane is an important concept for working in 3D. This section covers all aspects of the working plane, including setting, manipulating, and saving the working plane position.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

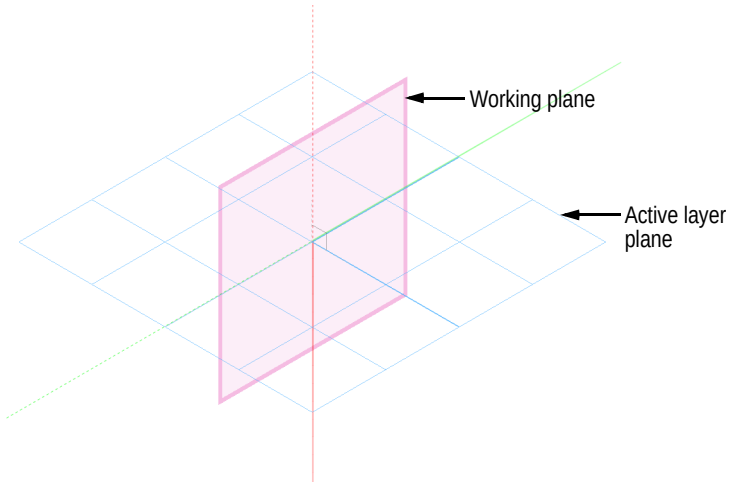
Understanding the Working Plane
The Active Planes List

Understanding the Working Plane

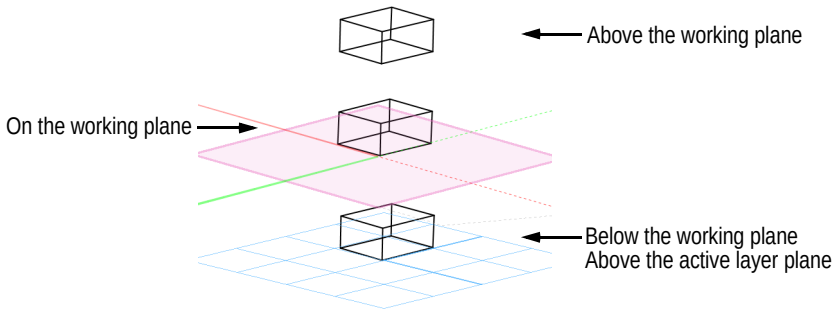
Every Vectorworks layer has a 3D plane associated with it. When a layer is active, this 3D plane is called the active layer plane. The active layer plane provides a constant visual and logical reference and is fixed in relation to the objects in the layer. In an architectural sense, the active layer plane is like the floor of a building; it is sometimes referred to as the ground plane.

Every Vectorworks drawing also has a working plane. The location of the working plane defaults to that of the active layer plane, and it is not seen. However, when working and modeling in 3D, the orientation of the working plane can be changed, and its positions saved. The working plane can be moved, aligned to different objects or surfaces, and rotated, unlike the active layer plane, which remains constant at the internal drawing center.

Working in 3D requires the cursor to snap to a 3D location. When no other 3D snap is active (the cursor is not snapping to 3D geometry), the cursor still needs to snap to a 3D location; this location is the 3D plane called the working plane.

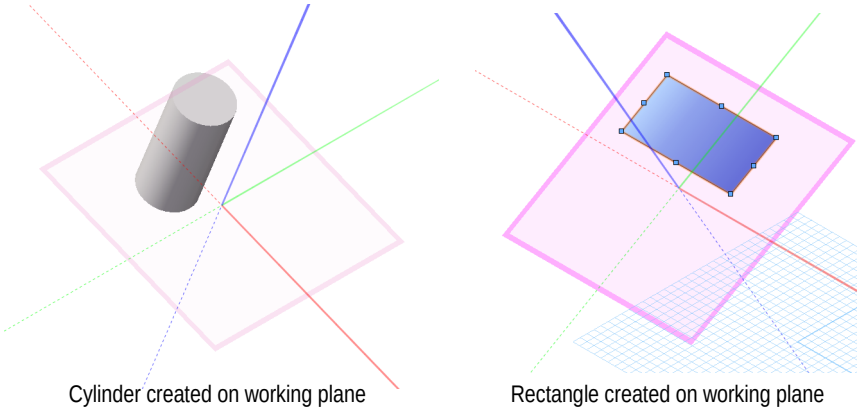


Every object created in the drawing is placed in relation to the active layer plane for that layer, whether it is placed directly on that plane or placed above or below it. The working plane helps to easily create and position objects in 3D space. It allows you to draw while in an isometric view and position objects exactly where they are needed. When changing the location of the working plane, the active layer plane remains displayed, providing a constant frame of reference while adjusting the working plane.



For example, for a drawing of a complex machine, with numerous gears, cams, rollers, and other parts that intersect on multiple planes, the working plane can be changed to different locations and angles in relation to the active layer plane.

Creating or locating objects according to the working plane allows highly accurate positioning of the objects in 3D space.



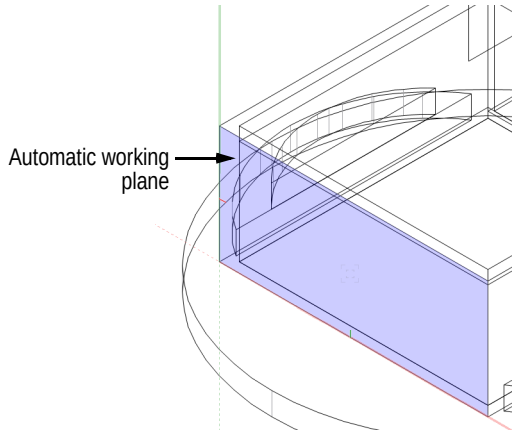
Cylinder created on working plane

Rectangle created on working plane

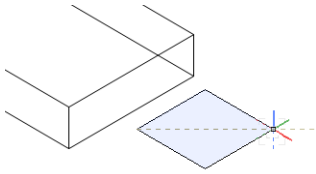
Hybrid symbols can only be inserted on a working plane that is parallel to the active layer plane. When inserting a hybrid symbol, if the working plane is not parallel to the active layer plane, the working plane is moved to, and aligned with, the active layer plane.

In addition, some tools require a vector or plane. The 3D viewing tools can use the working plane to define their center and/or axis of rotation, while the **Mirror** tool can mirror the selected 3D objects across the working plane. The location of 3D tools in 3D space is taken from the point on the working plane directly behind the cursor, or from a snap point if they are snapped to an object.

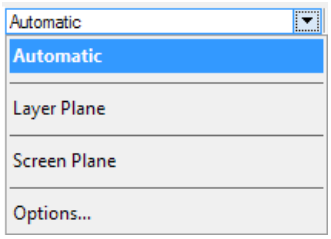
To assist with drawing in 3D with certain drawing tools, an automatic working plane appears on suitable object surfaces as the cursor moves over drawing objects with the tool selected. The automatic working plane is a temporary working plane that does not need to be explicitly set. Objects created on the automatic working plane are drawn planar to the automatic working plane, rather than on the layer plane. This facilitates drawing planar objects without the need to create or save a working plane. The color and opacity of the automatic working plane can also be customized; see “Configuring Interactive Display” on page 116.



If there is no suitable surface under the cursor, the automatic working plane aligns to the layer plane.



When the automatic working plane is in effect, Automatic displays in the Active Planes list located on the View bar.



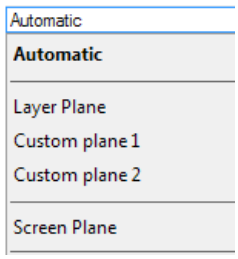
When drawing on a specific working plane that was established by the **Set Working Plane** tool, the automatic working plane should be turned off. Select a different plane from the list, such as a saved working plane, or press the \ key (backslash) to toggle the automatic working plane on and off. (This key can be customized in the Workspace Editor; see “Modifying Snapping and Mode Shortcuts” on page 1842.) When using a working plane tool such as the **Set Working Plane** tool, the automatic working plane defaults to off.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Appearance of the Planes
The Active Planes List
Setting the Working Plane Location
Manipulating the Working Plane
The Working Planes Palette
Aligning Objects to the Working Plane
The Automatic Working Plane

The Active Planes List

On the View bar, the Active Planes list displays the currently active plane, and also allows other active planes to be selected. The available planes depend on the drawing view, current tool, presence of named working planes, and plane mode options.



Plane	Description
Automatic	The active working plane changes as the cursor moves over the drawing area in a 3D view. The automatic plane is a temporary working plane that does not need to be set; over suitable surfaces and with certain drawing tools, the automatic plane is planar to the surface. Otherwise, the automatic plane aligns to the layer plane.
Layer Plane	The active working plane is planar to the current plane of the active layer
Custom Plane	The active working plane is planar to a saved working plane position; selecting a working plane cancels the automatic working plane.
Screen Plane	The active working plane is aligned to the screen plane (planar to the computer screen)
Options	Opens the Document Preferences dialog box, for setting the plane mode options; see “Plane Mode Preferences” on page 63

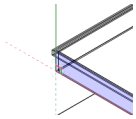
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Understanding the Working Plane

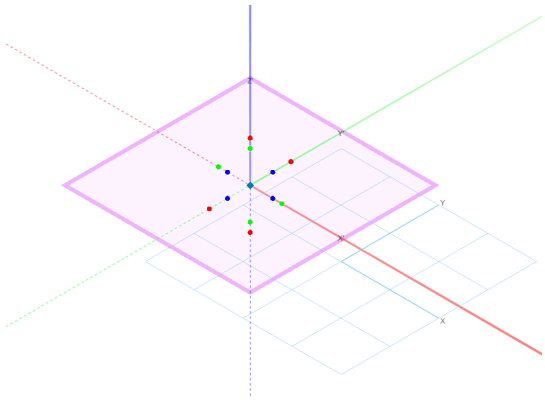
Appearance of the Planes

In any view other than Top/Plan, the layer plane is represented in blue; when active, it has red X and green Y axes (an option in the display preferences of the Vectorworks preference can also show the axes in Top/Plan). When the working plane and the active layer plane are located in the same position, only the active layer plane is shown. However, once the position of the working plane has been changed or directly set, it is represented on the screen as a colored frame. The color of the active layer plane and appearance of the working plane can be customized in the Vectorworks preferences; see “Interactive Preferences” on page 56 for details.

The axis lines of both the working plane and the automatic working plane are colored red for X’, green for Y’, and blue for Z’; dashed axes represent the negative axes. In a 3D view, the axes are always visible, although they appear more transparent when they are behind rendered solid objects. The Z’ axis and all axis labels are optionally displayed by settings in the SmartCursor Settings dialog box; see “Grid Snapping” on page 133.



Simply click on an axis with the **Set Working Plane** tool to select and display the working plane and its grip handles. When the working plane is selected, it displays with colored grips around each axis, as well as a move grip, for manipulating the plane.

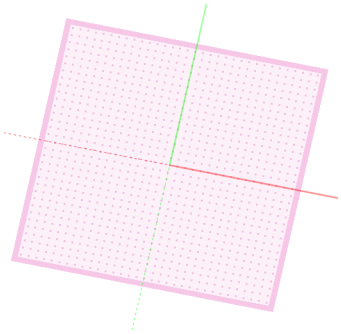


The working plane can be moved, but not rotated, when in rotated top/plan view (Vectorworks Design Series required). See “Rotating the Plan” on page 1163.

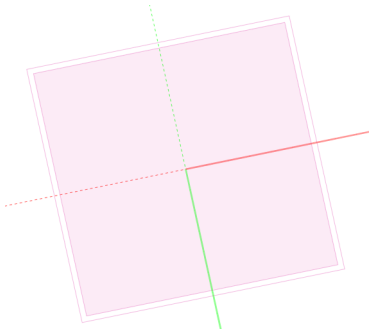
When grid snapping is enabled, the working plane displays with dots; when grid snapping is off, the dots do not display. The working plane displays slightly darker, and with a double line frame, when it is viewed from below.

When the working plane and the active layer plane are coincident, the active layer plane displays with grid lines. Otherwise, when the planes are not coincident, the grid lines do not display on the active layer plane.

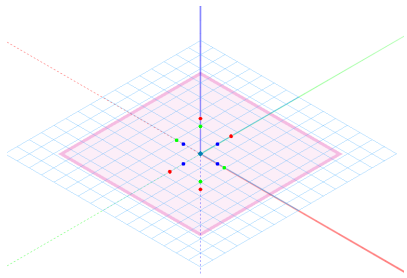
Dashed gray lines represent the intersection of the active layer plane and working planes. The active layer plane does not display in a rendered view.



Working plane with grid snapping enabled



Working plane seen from below



Active layer plane and working plane coincident, and working plane selected; Z axis display enabled

If the active layer plane grid lines are not visible, select **Show Grid** in the SmartCursor Settings dialog box (see "Grid Snapping" on page 133).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Working Plane Location

The **Set Working Plane** tool can set the location and angle of the working plane. Set the working plane to any location in 3D space.

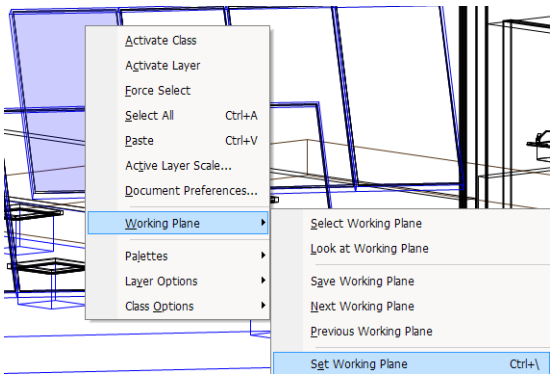
Double-click on the **Set Working Plane** tool to set the working plane to the layer plane. Double-click on a planar object to re-activate the working plane on which it was created.

The **Set Working Plane** tool has two modes:

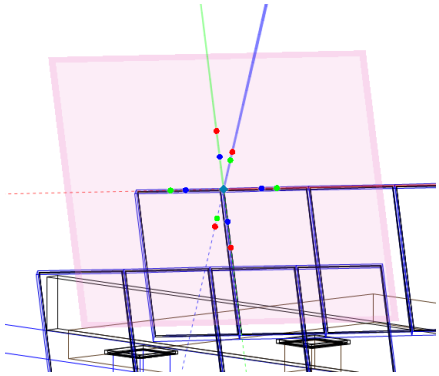
Mode	Description
Three Point	Defines the working plane by three, 3D points, or according to the surface of a rendered object
Planar Face	Aligns the working plane to a planar face; or, for a NURBS curve, aligns the working plane perpendicular to that point's tangent on the curve

Once the working plane location has been set, Custom Plane displays in the Active Planes list on the View bar. The location can be saved; see "Working Plane Commands" on page 1174.

The working plane can be quickly set based on the automatic working plane location. Right-click (Windows) or Ctrl-click (Mac) on the automatic working plane, and select **Set Working Plane** from the context menu (or select **Modify > Working Plane > Set Working Plane**). The working plane is set to match the automatic working plane, and the working plane is active.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



With the automatic working plane in the desired location, select **Set Working Plane**



The working plane is active, set to match the automatic plane

Click here for a video tip on this topic (Internet connection required).

Setting the Working Plane with Three Points
Setting the Working Plane to a Planar Face
Manipulating the Working Plane

Setting the Working Plane with Three Points

Two methods are available for defining the working plane location in Three Point mode. The first method allows the position and angle of the working plane to be set by defining the origin and axes of the new working plane location. The second method sets the position and angle of the working plane according to a rendered object's surface.

Setting the Working Plane

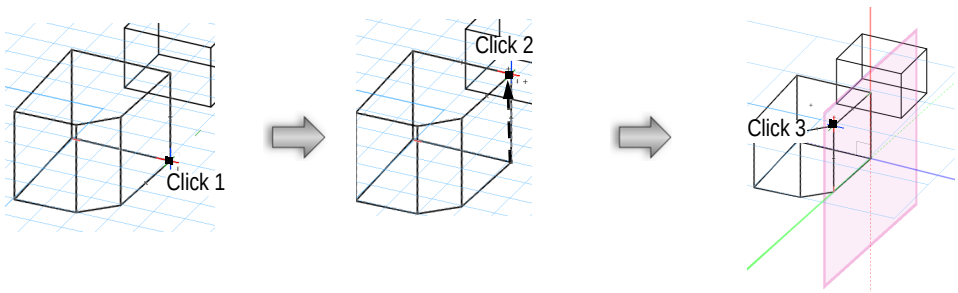
To set the working plane with three points:

1. In a 3D view, click the **Set Working Plane** tool from the 3D Modeling tool set, and then select **Three Point** from the Tool bar.
2. Click to set the first point.

   This becomes the origin of the working plane.
3. Click to set the second point.

   This sets the working plane X' axis.
4. Select the third point.

   This sets the Y' axis for the working plane. Once the third point is defined, the working plane location is set.



Setting the Working Plane to a Rendered Object's Surface

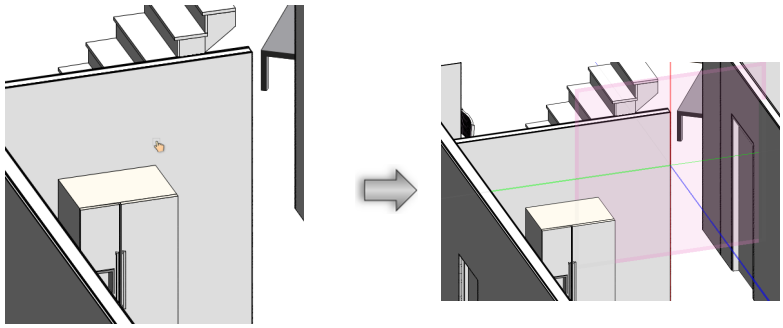
The working plane can be defined according to the surface of a rendered object.

To set the working plane to a rendered object's surface:

1. In a 3D view, select a rendering mode of Unshaded Polygon, Shaded Polygon, Shaded Polygon No Lines, or Final Shaded Polygon.
2. Click the **Set Working Plane** tool from the 3D Modeling tool set, and then select **Three Point** from the Tool bar.

   The cursor changes to a pointing hand when over the surface of a rendered object.
3. Click on the surface that defines the working plane location.

   The working plane location is set to the selected surface.



Click the rendered surface to use for aligning the working plane

Setting the Working Plane Location
Understanding the Working Plane
Manipulating the Working Plane
The Working Planes Palette
Aligning Objects to the Working Plane

Setting the Working Plane to a Planar Face

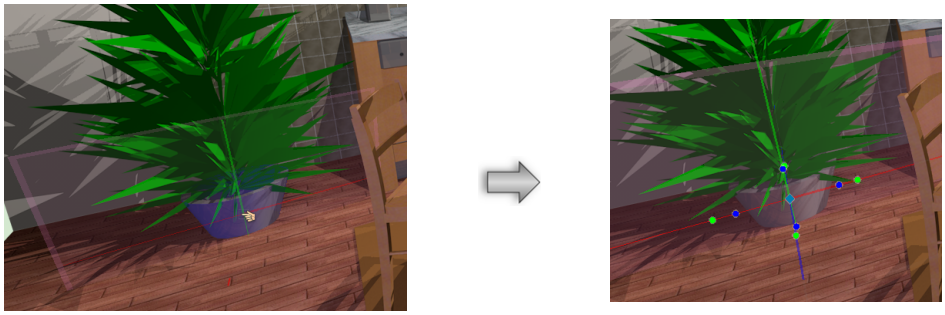
Planar Face mode is particularly useful when aligning the working plane to the top of an object without sharp corners (like a cylinder); instead of clicking in three places, only the planar face of the object needs to be selected.

In addition, the working plane can be positioned on a NURBS curve, aligning the working plane's Z axis to the curve's tangent direction. The ability to set the working plane to a NURBS curve requires one of the following conditions: a smart point must be set on the NURBS curve, **Snap to Object** must be enabled, or the **Nearest Point on Edge** object snapping must be enabled.

To set the working plane to a planar face:

1. In a 3D view, click the **Set Working Plane** tool from the 3D Modeling tool set, and then select **Planar Face** from the Tool bar.
2. As the cursor moves over the underlying geometry, a preview of the working plane displays. Click to set the working plane location. The click location becomes the origin of the working plane.

   The working plane is aligned to the selected face and remains selected for manipulation. The working plane is deselected when the **Set Working Plane** tool is no longer active.



Manipulating the Working Plane

Working plane operations can be performed either by using the grips on the selected plane or by menu commands.

Working Plane Grips

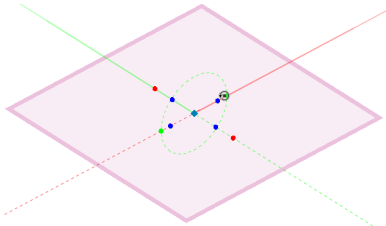
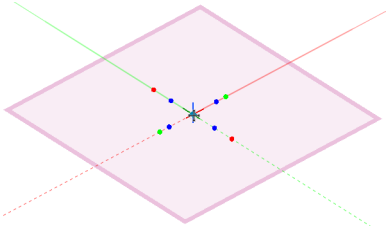
Manipulate the working plane by rotating about any of the three axes (X', Y', and Z') or moving the working plane origin.

To manipulate the working plane directly:

1. Click the **Selection** tool from the Basic palette.
2. In a 3D view, select the working plane by clicking on its edge or any of its axes. Alternatively, double-click on a planar object to re-activate the working plane on which it was created.

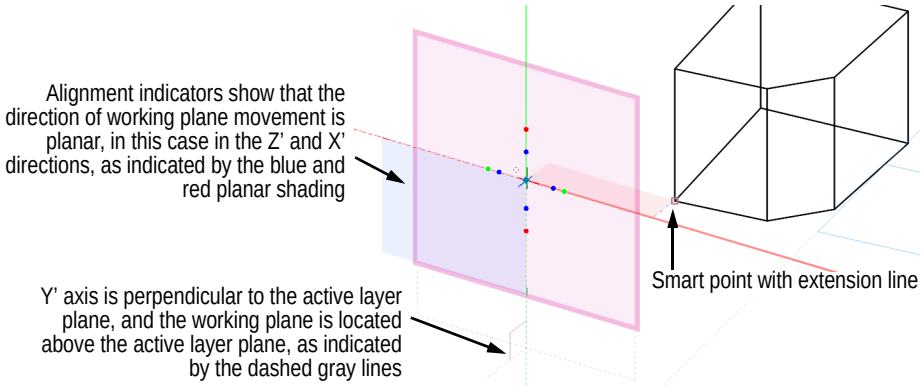
   The working plane must be selected for the grips to be visible. Any other object reshape handles are temporarily hidden, to avoid confusion.
3. The working plane displays with grips.

   Each axis has two grips for rotation about the other two axes. Red grips rotate about the X' axis, green grips rotate about the Y' axis, and blue grips rotate about the Z' axis. Moving the cursor over an axis displays a preview showing the rotation direction. The blue diamond grip at the working plane origin moves the working plane.

Action	Description
Rotate the working plane	Click on one of the axis grips. The rotate cursor displays. Move the cursor to rotate the plane about the selected axis, and click to set. 
Move the working plane	Click on the center grip. The move cursor displays. Move the cursor to move the plane, and click to set. 

4. Once the working plane has been positioned, press the Esc key, or click off of the working plane. The position can be saved with working plane commands or in the Working Planes palette.

   Use the grips in combination with SmartCursor cues for accurate manipulation in relation to other objects.



The working plane can be moved, but not rotated, when in rotated plan view (Vectorworks Design Series required). See “Rotating the Plan” on page 1163.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Working Plane Commands
The Working Planes Palette

Working Plane Commands

The working plane commands can select and manipulate the working plane. Some commands have equivalents in the Working Planes palette or when editing the working plane directly.

To access the working plane commands:

Select **Modify > Working Plane**.

The same commands are available from the context menu of the working plane.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Command	Description
Select Working Plane	Selects the working plane (alternatively, click on the plane with the **Selection** tool, select **Edit > Select Working Plane**, or select the working plane from the Working Planes palette's **Utility** menu or context menu)
Look at Working Plane	Changes the view to be perpendicular to the working plane; similar to the **Top** view under the **View** menu, in that you are looking straight at the working plane. This command is also available from the **View** menu and the View bar.
Save Working Plane	Opens the New Working Plane dialog box. Enter a name to save the working plane position; if desired, change the origin and rotation values. Once a working plane position has been saved, it can be accessed through the Working Planes palette and the Active Planes list on the View bar. Working planes can also be saved by selecting **New** from the Working Planes palette's **Utility** menu or context menu.
Next Working Plane	Cycles through the next ten unsaved palette positions
Previous Working Plane	Cycles through the last ten unsaved palette positions
Set Working Plane	If selected while the automatic working plane is highlighted, sets the working plane to match the automatic working plane. Otherwise, sets the working plane to the next clicked planar face, similar to the Planar Face mode of the **Set Working Plane** tool (see "Setting the Working Plane to a Planar Face" on page 1172)
Align Working Plane X Axis with Layer Plane	Rotates the working plane coordinates about the Z axis until the X axis is parallel to the active layer plane and the Y axis is pointing up. The working plane origin is unchanged.
Align Working Plane with Layer Plane	Aligns all working plane axes to the corresponding active layer plane axes. The working plane origin is unchanged.
Align Working Plane with Current View	Aligns the working plane Z axis toward the viewer; the X axis is aligned horizontally and the Y axis vertically, on the screen. The working plane origin is unchanged.
Rotate About X' Left 90°	Rotates the working plane to the left about its X axis by 90 degrees
Rotate About X' Right 90°	Rotates the working plane to the right about its X axis by 90 degrees
Rotate About Y' Left 90°	Rotates the working plane to the left about its Y axis by 90 degrees
Rotate About Y' Right 90°	Rotates the working plane to the right about its Y axis by 90 degrees
Rotate About Z' Left 90°	Rotates the working plane to the left about its Z axis by 90 degrees
Rotate About Z' Right 90°	Rotates the working plane to the right about its Z axis by 90 degrees
Flip X'	Flips the working plane about its X axis
Flip Y'	Flips the working plane about its Y axis
Flip Z'	Flips the working plane about its Z axis

When using the rotate commands, left indicates counterclockwise when viewing the working plane origin from the positive X axis.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Working Plane Grips
The Working Planes Palette
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

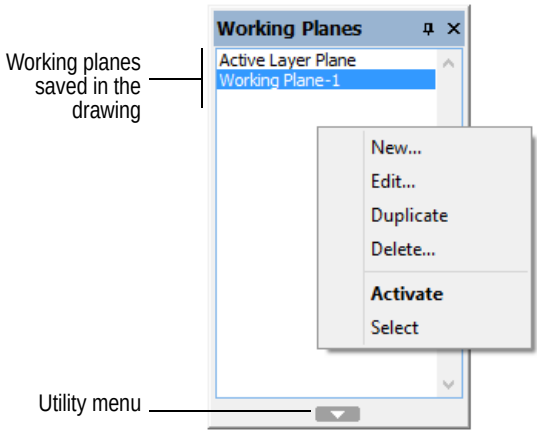
The Working Planes Palette

The Working Planes palette displays working planes that have been set or saved while working on a drawing. The palette allows you to save new working plane positions, and to edit, duplicate, delete, activate, and select existing working planes. Combine these features with the working plane controls on the View bar, the working plane commands on the **Modify** menu, the **Set Working Plane** tool, and the **Align Plane** tool to accurately control the placement and alignment of objects in 3D space.

To manage working planes in the Working Planes palette:

1. Select **Window > Palettes > Working Planes**.
2. The Working Planes palette opens. Click the **Utility Menu** button at the bottom of the palette to open the **Utility** menu.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the palette to access the context menu.



Menu/Command	Action
New	Saves a new working plane; see “Saving Working Plane Positions” on page 1177
Edit	Edits the currently selected working plane; see “Editing and Renaming Working Planes” on page 1177
Duplicate	Duplicates all currently selected working planes
Delete	Deletes all currently selected working planes
Activate	Activates the currently selected working plane. Alternatively, double-click on an existing working plane from the Working Planes palette to activate it.
Select	Activates the currently selected working plane, if it is not already active, then performs the **Select Working Plane** command to display the grip handles for the plane

3. Select the desired command.

Saving Working Plane Positions
Editing and Renaming Working Planes
Accessing Existing Working Planes
Working Plane View and Modes
Working Plane Commands

The Active Planes List

Saving Working Plane Positions

The working plane position can be permanently saved with the drawing and accessed later from the Working Planes palette or Active Planes list on the View bar.

To permanently save a working plane position:

1. Select **Window > Palettes > Working Planes**.

   The Working Planes palette opens.

2. Click the **Utility Menu** button at the bottom of the palette to open the **Utility** menu.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the palette to access the context menu.

3. Select **New**.

   Alternatively, select **Modify > Working Plane > Save Working Plane** or right-click (Windows) or Ctrl-click (Mac) on the drawing to access the context menu.

   The New Working Plane dialog box opens. If there is a currently active working plane, the dialog box displays the origin and rotation values of the current plane. Otherwise, the dialog box displays the origin and rotation values of the global ground plane centered at the internal origin.

4. Name the working plane position and reset the values, if desired.

5. Click **OK**. Saved working plane positions are displayed in alphabetical order in the palette. To access a saved working plane position, double-click its name in the Working Planes palette or select it from the Active Planes list on the View bar.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Working Planes Palette
Editing and Renaming Working Planes

Editing and Renaming Working Planes

To edit or rename a working plane position:

1. Select **Window > Palettes > Working Planes**.

   The Working Planes palette opens.

2. Select the working plan to edit.

3. Click the **Utility Menu** button at the bottom of the palette to open the **Utility** menu.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the palette to access the context menu.

4. Select **Edit**.

   The Edit Working Plane dialog box opens.

5. Enter the new name and/or origin and rotation values.

6. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Working Planes Palette

Accessing Existing Working Planes

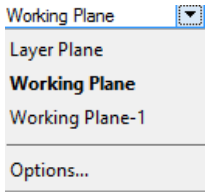
Initially, when the working plane has not yet been changed, the only plane saved in the Working Planes palette is that of the active layer plane location, and only the active layer plane or the automatic working plane are available from the Active Planes list in the View bar. Once the position of the working plane has been changed, it can be accessed from different locations depending on whether the position has been saved.

Accessing Saved Working Planes

The Active Planes list on the View bar displays and switches among saved working planes. See “The View Bar” on page 37 and “The Active Planes List” on page 1168. Selecting a working plane cancels the automatic working plane.

To access working planes from the View bar:

While in a view other than Top/Plan, select a working plane from the Active Planes list.



The current working plane displays in bold; if it has not yet been saved, it displays as Working Plane.

Alternatively, select **Window > Palettes > Working Planes** and double-click the name of the plane from the Working Planes palette.

The selected working plane displays.

Accessing Unsaved Working Plane Positions from the Current Drawing Session

To access unsaved working plane positions, select **Modify > Working Plane > Next Working Plane** and **Modify > Working Plane > Previous Working Plane**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Working Planes Palette
The Active Planes List
Working Plane Commands

Working Plane View and Modes

The three buttons on the View bar control how the working plane is viewed and how it interacts with certain tools and commands.

Button	Description
Look At Working Plane 	Changes the view to be perpendicular to the working plane; similar to the **Top** view under the **View** menu in that you are looking straight at the working plane
Active Layer Plane Views 	Sets the active layer plane as the reference for tools such as the **Flyover** tool, **Walkthrough** tool, and other viewing tools; used with the **Flyover** tool’s rotation center mode
Working Plane Views 	Sets the working plane as the reference for tools such as the **Flyover** tool, **Walkthrough** tool, and other viewing tools; used with the **Flyover** tool’s rotation center mode

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Working Planes Palette
Working Plane Commands

Aligning Objects to the Working Plane

When the working plane has been set to the desired location, it can be used to position 3D objects. One way to align objects precisely in 3D space is to align them to the working plane.

Click here for a video tip about this topic (Internet access required).

Setting Objects to the Working Plane with Three Points
Setting an Object Surface to the Working Plane

Setting Objects to the Working Plane with Three Points

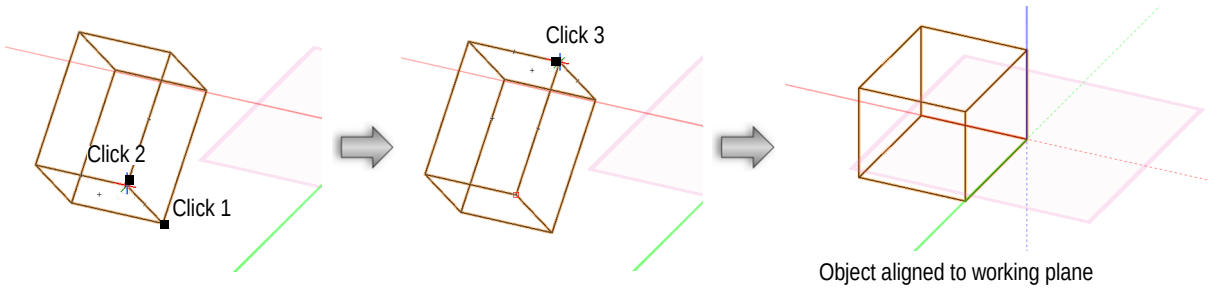
To align an object to the working plane with three points:

1. In a 3D view, set the working plane location as desired.
2. Select the object to align to the working plane.
3. Click the **Align Plane** tool from the 3D Modeling tool set.
4. Click a location on the object to align with the origin of the working plane.
5. Click a second point on the object to define the X axis.

   The line between the first and second points defines the X axis position.
6. Click a third point on the object to define the Y axis.

   The line between the second and third points defines the Y axis position.

   The object aligns itself to the working plane as defined by these points. If the surface was mistakenly aligned to the wrong side of the working plane, use the **Mirror** tool to flip the object to the opposite side of the working plane.



Setting an Object Surface to the Working Plane

Setting an Object Surface to the Working Plane

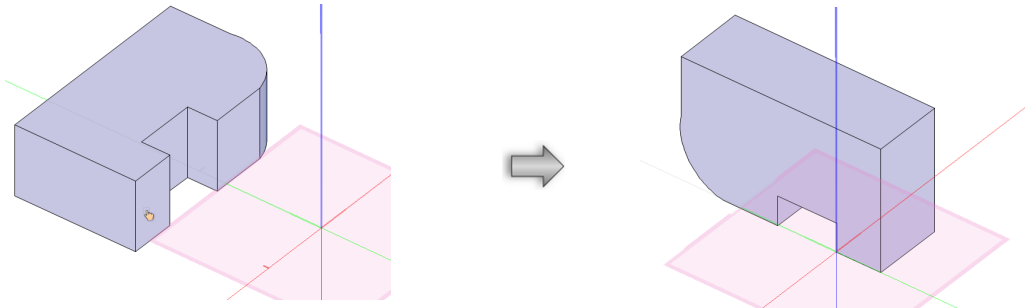
A rendered object can be aligned to the working plane.

To align a rendered object to the working plane:

1. Select a 3D view and a rendering mode for the drawing.

   The drawing projection must be other than Top/Plan, with a rendering mode of Unshaded Polygon, Shaded Polygon, Shaded Polygon No Lines, or Final Shaded Polygon.
2. Select the 3D object to align to the working plane.
3. Click the **Align Plane** tool from the 3D Modeling tool set.

   The cursor changes to a pointing hand when over the surface of a rendered object.
4. Click on the surface to align to the working plane.



Click the rendered surface to align to the working plane

The object's selected surface is aligned to the working plane.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Objects to the Working Plane with Three Points

Animating Drawings

Two types of animations can be created from a Vectorworks 3D drawing—orbit point and move along path. The orbit point animator rotates by a specified number of degrees around a 3D object or selected point in the drawing. The move along path animator moves through the 3D drawing, following a specified path.

In Perspective view, only the portion of the model within the perspective frame (see "Cropped and Uncropped Perspective Views" on page 1144) is visible in the animation.

Windows-generated and Mac-generated movies can be viewed on either platform. Simply double-click the file to open it, or open it from your preferred viewer.

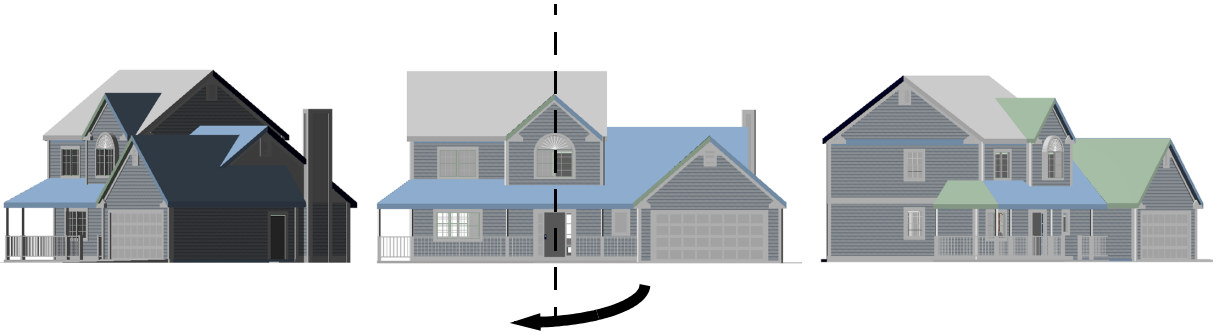
> On Mac and Windows 7/8 platforms, an .mov (MPEG4) file with H.264 compression is created; on Windows XP and Vista an avi file with optional MJPEG compression is created.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Orbit Point Animations
Creating and Editing Move Along Path Animations
Adding Text or Title Screens to Animations

Creating Orbit Point Animations

The orbit point animator creates an animation that moves in a circular path around a specific 3D object or point.



To create an orbit point animation:

1. Set up the drawing view.

   Select the desired views from the **View** menu—**Standard Views**, **Projection**, and **Rendering**. In addition, use the **Zoom** tool to set the drawing magnification level. Ensure that only the layers and classes that should display in the animation are visible.

2. To specify an object or objects as the center of rotation, select the object or group of objects.
3. Select **Model > Create Animation**.

   The Create Animation dialog box opens.

4. Set the **Camera** to Orbit Point.
5. Click **Animation Options**.

   The Orbit Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Rotation angle	Specifies the amount of rotation (in degrees) for the animation; for example, to complete an orbit around the selected center of animation, enter 360
Rotate about center of	
Active Layer Plane	Specifies the center of the active layer plane (0X, 0Y, 0Z) as the center of rotation for the animation
Working Plane	Specifies the center of the working plane (0I, 0J, 0K) as the center of rotation for the animation
Selection	Specifies the center of the selected object(s) as the center of rotation for the animation

6. Click **OK** to return to the Create Animation dialog box.
7. Specify the remaining parameter settings in the Create Animation dialog box.

Click to show/hide the parameters.

Parameter	Description
Duration (in seconds)	Sets the total length of time of the animation

Parameter	Description
Time Scale	Specifies the animation rate; a value between 0.1 and 0.99 creates a slow motion effect, while a value between 1.01 and 10.00 speeds up the animation. Leave the default value of 1.00 for a normal time scale.
Frames per second	Sets the number of frames per second (fps) in the animation. 30 fps generally produces a smooth final animation.
Quality	Sets the quality of the movie output; the higher the quality, the larger the resulting file. The Very High setting is of appropriate quality for follow-on video editing.
Time Range	Sets the duration of a preview or saved movie relative to the full movie duration.
Full Duration	Previews/saves the complete movie
Limited Duration	Previews/saves only the specified section of the movie. **Start Time** and **End Time** can be set to any value between 0 and the **Duration** of the movie.

8. Click **Preview** to check the animation before saving it. The preview defined by the **Time Range** settings is shown in the current interactive render mode (OpenGL or Wireframe).

   At the end of the preview, the view is returned to the view displayed before the preview began.

9. When satisfied with the preview, click **Save Movie** to save the animation.

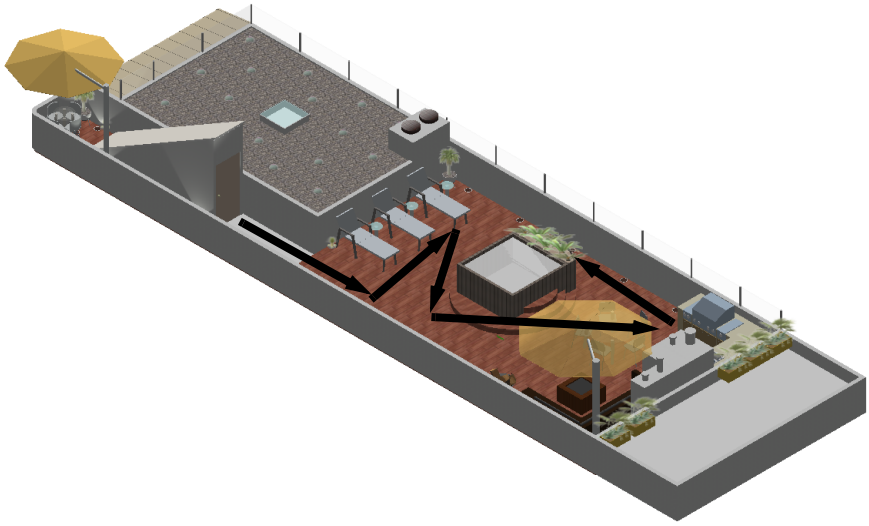
   The saved movie observes the current **Time Range** settings.

   The Save As dialog box opens.

10. Enter the name for the movie file and specify its location. Click **Save**. The progress of movie creation is displayed.

Creating and Editing Move Along Path Animations

The move along path type of animation moves through a 3D drawing along a specified path. For example, create a walk-through presentation of a house.



To create an animation moving along a specified path:

1. Set up the drawing view.

   Select the desired views from the **View** menu: **Standard Views**, **Rendering**, and **Projection**.

The drawing **Projection** must be set to a perspective view.

2. Save a view for each point along the path to use for creating the animation.

   Use the **Walkthrough** and/or **Flyover** tools to change views. To save a view, select **View > Save View**. In the Save View dialog box that opens, enter the **View Name**, select the view parameters for saving, and then click **OK** (see "Creating Saved Views" on page 189).

3. Select **Model > Create Animation**.

   The Create Animation dialog box opens.

4. Set the **Camera** to Move Along Path.

5. Click **Animation Options**.

   The Select Animation dialog box opens.

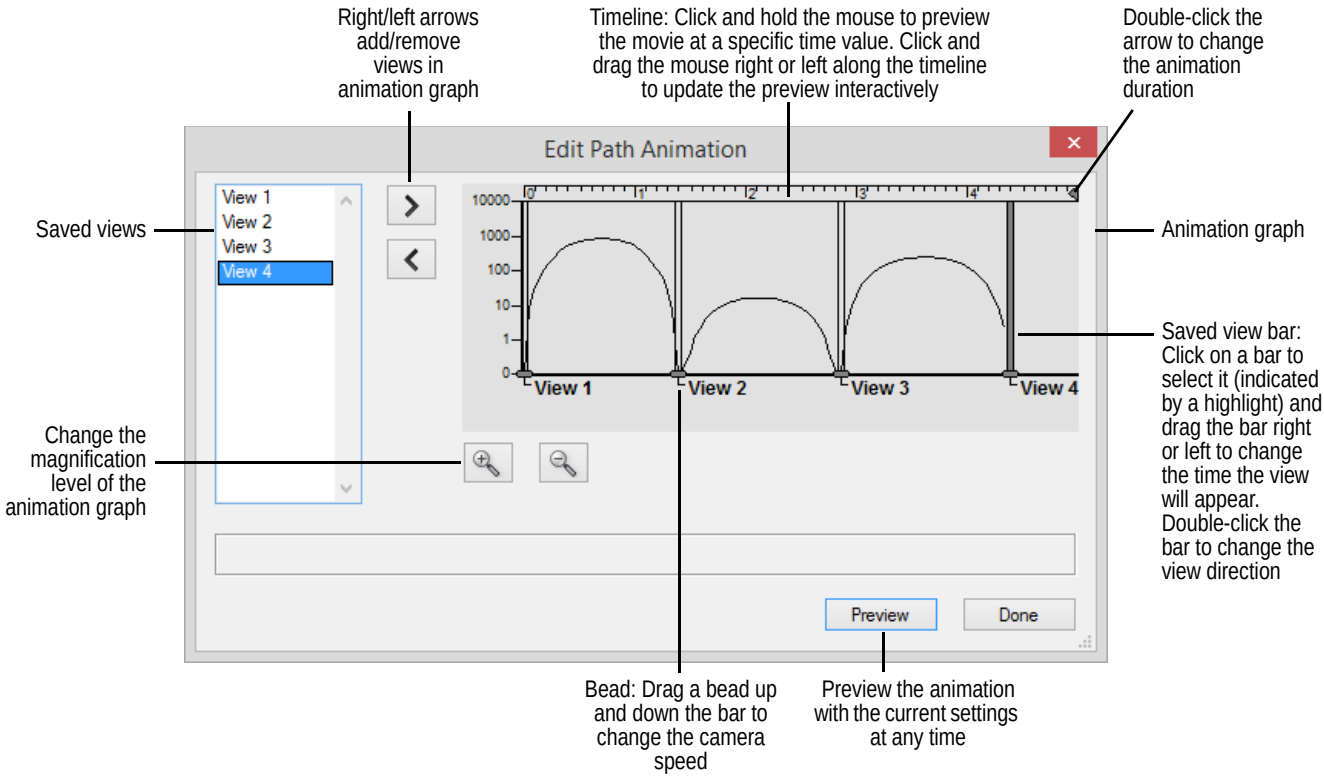
6. Click **New**.

   The New Animation Name dialog box opens.

   Enter a name for this animation and then click **Create** to return to the Select Animation dialog box.

7. Select the name of the animation to edit and click **Edit**.

   The Edit Path Animation dialog box opens.



8. From the list of views on the left, select the starting view name and click the right arrow to add it to the animation graph.

   The first view is placed at the graph's origin: 0 seconds, 0 drawing units/second.

9. Select the next view to use and click the right arrow to add it to the animation graph. Continue selecting views and adding them to the animation graph until the desired views have been included within the time allotment.

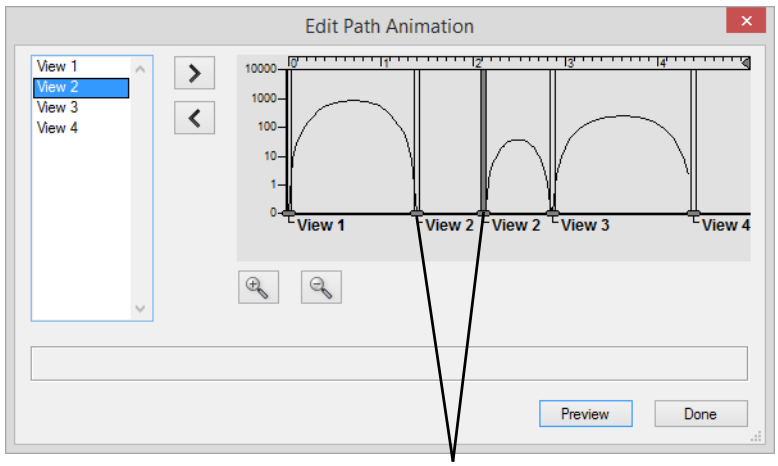
10. For each added view, a bar with a bead is added to the graph. Drag the bar and bead to change the animation settings (time elapsed between view changes and velocity of camera movement).

The distance between bars is the time in seconds that it takes to move from one view to the next.

Move the bead up and down to determine the slope of the line between bars. This slope indicates the velocity of the movement between views (the number of drawing units/second that the camera moves). In general, the slope should form a steady curve. An uneven curve will cause a choppy camera movement, speeding up and slowing down in a jolting manner. The slope may dip below the X axis; however, this may result in a negative velocity.

If a bar is currently selected, the saved view is inserted between that bar and the following bar. If no bars are selected, the saved view is inserted after all the bars that are currently in the animation graph. Bars can be dragged past each other to change the order of the views' appearance. Each saved view can be added to the graph multiple times.

To add a pause to the animation, add the same view twice to the animation graph so that the views are next to each other in sequence. The line between the views should be flat (no upward or downward slope). This creates a velocity of zero and, therefore, a pause in the animation.



To create a pause, place the same view twice in the animation graph, with no slope between the repeated views

Setting the view to different specified X, Y, Z coordinates for each of these frames, makes the camera appear to stop its forward motion and pan from one direction to another.

11. To set a specific camera target, double-click on a bar.

    The Set View Direction dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Look towards	Specifies the direction toward which the view should look
Current view	Sets the view to the current bead's saved view
Working plane origin	Sets the view to the working plane origin
Center of currently selected object	Sets the view to the X, Y, Z center of the currently selected object(s) bounding box
Center of following named object	Sets the view to look at the center of a named 3D object. Click **Select a different named object** to open the Look At Named Object dialog box. Select the object to look toward (named objects in walls or layer links cannot be selected).
Following point	Sets the view to the specified X, Y, Z coordinates

12. Click **OK** to exit the **Set View Direction** dialog box.
13. Click **Done** to return to the Create Animation dialog box.
14. Specify the remaining parameter settings in the Create Animation dialog box.

Click to show/hide the parameters.

Parameter	Description
Duration (in seconds)	Sets the total length of time of the animation. This value is automatically set to five seconds for move along path animations, and this field is disabled to prevent accidental truncation of the specified path. The duration can be changed by double-clicking the arrow to the far right of the timeline on the Edit Path Animation dialog box.
Time Scale	Specifies the animation rate; a value between 0.1 and 0.99 creates a slow motion effect, while a value between 1.01 and 10.00 speeds up the animation. Leave the default value of 1.00 for a normal time scale.
Frames per second	Sets the number of frames per second (fps) in the animation. 30 fps generally produces a smooth final animation.
Quality	Sets the quality of the movie output; the higher the quality, the larger the resulting file. The Very High setting is of appropriate quality for follow-on video editing.
Time Range	Sets the duration of a preview or saved movie relative to the full movie duration.
Full Duration	Previews/saves the complete movie
Limited Duration	Previews/saves only the specified section of the movie. **Start Time** and **End Time** can be set to any value between 0 and the **Duration** of the movie.

15. Click **Preview** to check the animation before saving it. The preview defined by the **Time Range** settings is shown in the current interactive render mode (OpenGL or Wireframe).

    At the end of the preview, the view is returned to the view displayed before the preview began.
16. When satisfied with the preview, click **Save Movie** to save the animation.

    The saved movie observes the current **Time Range** settings.

    The Save As dialog box opens.
17. Enter the name for the movie file and specify its location. Click **Save**. The progress of movie creation is displayed.

Duplicating and Deleting Move Along Path Animations

Existing move along path animations can be duplicated or deleted from the Select Animation dialog box.

To duplicate an animation:

1. Select the animation name from the animations list and click **Duplicate**.
2. Enter a name for the duplicate animation, and click **Create**.

   The duplicate animation is created.

To delete an animation:

1. Select the animation name from the animations list and click **Delete**.
2. Click **OK**.

   The animation is deleted.

Adding Text or Title Screens to Animations

Unless text is specifically converted into a 3D object (using the **Convert Text to Polylines** command), the Vectorworks program views text as 2D. This means text in a drawing exported as a movie remains motionless, not moving with the other 3D objects. This principle is also true for any graphic images placed in the drawing.

The Vectorworks program can be used to create an animated title screen as a separate movie, using an orbit point animation to move the 2D text. (The move along path animator requires 3D perspective projection.) Then, using a third-party video editor, link the two files together to create one movie.

When creating a title screen animation, ensure that the title movie uses the same frame rate setting as the linked animation movie.

Dimensions

Use the various dimensioning tools to measure 2D and 3D objects, and to add dimensioning lines and measurements to the drawing. Dimensions can be created on a design layer, or in the annotation space of a sheet layer viewport (see “Creating Annotations for Sheet Layer Viewports” on page 1653).

There are also **Tape Measure** and **Protractor** tools to measure distances and angles, and a **Center Mark** tool to mark the center of circles, ovals, and rectangles.

When you add a new dimension, the dimension standard set in the document preferences is used by default (see “Dimension Preferences” on page 61). In addition, when a dimension tool is active, the default dimension standard can be set from the Tool bar. Select any of the built-in dimension standards, or create a custom standard specifically for the drawing. If the default dimension standard for the document is changed, any new dimension that is added will use the new standard; no existing dimensions are affected.

~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Custom Dimension Standards

Custom dimension standards can be created in the current file or imported from another drawing file. There are three ways to access the Custom Dimensions dialog box for managing custom dimensions.

- Select **File > Document Settings > Document Preferences**. From the Document Preferences dialog box, click the Dimensions tab, and then click **Custom**.

  Alternatively, access the **Document Preferences** command from the document context menu (right-click on Windows, or Ctrl-click on Mac), or from the **Quick Preferences** menu on the Tool bar.

- With any one of the dimension tools in the Dims/Notes tool set selected, open the **Dim Std** list on the Tool bar and select Custom Standards.
- With an existing dimension selected, open the **Dim Std** list on the Object Info palette and select Custom Standards.

From the Custom Dimensions dialog box, several functions are available.

Function	Description
New	Opens the Assign Name dialog box, to create a new custom dimension standard; see “Creating a Custom Dimension Standard” on page 1188
Delete	Deletes the selected custom dimension standard and replaces it with the standard you specify; see “Creating a Custom Dimension Standard” on page 1188
~~~~~~~~~~~~~~~~~~~~~~~~~~

Function	Description
Edit	Opens the Custom Dimension dialog box, to edit the selected custom dimension standard; see “Editing a Custom Dimension Standard” on page 1189
Rename	Opens the Assign Name dialog box, to rename the selected custom dimension standard; see “Renaming a Custom Dimension Standard” on page 1190
Import	Opens a file and imports the custom dimension standard you specify; see “Importing a Custom Dimension Standard” on page 1191
Replace	Opens the Replace Dimension Standards dialog box, to specify which dimension standard to replace, and which standard to replace it with; see “Replacing a Dimension Standard” on page 1191

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Custom Dimension Standard
Editing a Custom Dimension Standard
Deleting a Custom Dimension Standard
Renaming a Custom Dimension Standard
Importing a Custom Dimension Standard
Replacing a Dimension Standard
Dimensions

Creating a Custom Dimension Standard

To create a custom dimension standard:

1. Select **File > Document Settings > Document Preferences**. From the Document Preferences dialog box, click the Dimensions tab, and then click **Custom**.

   Alternatively, with any one of the dimension tools in the Dims/Notes tool set selected, open the **Dim Std** list on the Tool bar and select Custom Standards from the list. Alternatively, with an existing dimension selected, open the **Dim Std** list on the Object Info palette and select Custom Standards from the list.

   The Custom Dimensions dialog box opens.

2. Click **New**.

   The Assign Name dialog box opens.

3. Enter a name for this dimension standard and click **OK**.

4. Select the new dimension from the Standards list and click **Edit**.

   The Edit Custom Dimension Standard dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Dimension line distances	Specifies the dimension line distances for linear, radial, and ordinate dimensions
Note: Distances are in	Displays the drawing units currently in use for the file
Witness Lines	Applies witness lines to dimensions; when deselected, witness lines are hidden
Fixed Witness Line Length	Select to use a fixed length for the witness lines rather than a fixed offset from the dimensioned object (the default). When selected, the field for setting the witness line length is enabled on the top part of the dialog box. To change the length of a particular witness line, use the **Override** option in the Object Info palette
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Dimensions Always Inside	When selected, the dimension value is always centered between the witness lines. When deselected, the dimension value is automatically positioned outside the witness lines if it is too large to fit between them.
Linear Markers	Select the marker attributes for use with linear dimension markers (see "Marker Attributes" on page 1102)
Other Markers	Select the marker attributes for use with other dimension markers
Leader Line Markers	Select the marker attributes for use with dimension leader lines. The marker is only visible if you first use the Object Info palette to add a leader line to the dimension, and then use the Attributes palette to add an end marker to the leader line (see "Marker Attributes" on page 1102).
SIA Format Numbers	Sets all numbers to use the SIA format; not available when a dual layout is selected
Tolerance Size	Sets the size of the tolerance in relation to the dimension text
Text Rotation	Sets how text is handled when a dimension is rotated; text can be horizontal, aligned, or horizontal/vertical
Text Position	Sets the location of the dimension value relative to the dimension line: • Above/Left places the text above the dimension line for non-vertical dimensions, or left of the dimension line for vertical dimensions • Above/Right places the text above the dimension line for non-vertical dimensions, or right of the dimension line for vertical dimensions • Outside places the text outside the area enclosed by the two witness lines and dimension line, regardless of the dimension orientation
Layout	Sets whether dimensions display as single values, dual - side by side, or dual - stacked
Dual view	If a dual layout is selected, specify whether both dimensions are shown, primary only, or secondary only
Text Style	Select a text style from either the default content or the current file's content. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.

5. Enter the desired values for the display of dimensions, and then click **OK**. Click **OK** again to close the Custom Dimensions dialog box.
6. To set the new standard as the default for the document, select the new standard from the **Dimension Standard** list, and then click **OK**.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Custom Dimension Standards
Editing Dimension Properties

Editing a Custom Dimension Standard

To edit a custom dimension standard:

1. Select **File > Document Settings > Document Preferences**. From the Document Preferences dialog box, click the Dimensions tab, and then click **Custom**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Alternatively, with any one of the dimension tools in the Dims/Notes tool set selected, open the **Dim Std** list on the Tool bar and select Custom Standards. Alternatively, with an existing dimension selected, open the **Dim Std** list on the Object Info palette and select Custom Standards.

The Custom Dimensions dialog box opens.

2. Select the custom standard to edit, and click **Edit**.

   The Custom Dimension dialog box opens.

3. Edit the settings as needed, and click **OK**.

Using Custom Dimension Standards

Deleting a Custom Dimension Standard

To delete a custom dimension standard:

1. Select **File > Document Settings > Document Preferences**. From the Document Preferences dialog box, click the Dimensions tab, and then click **Custom**.

   Alternatively, with any one of the dimension tools in the Dims/Notes tool set selected, open the **Dim Std** list on the Tool bar and select Custom Standards. Alternatively, with an existing dimension selected, open the **Dim Std** list on the Object Info palette and select Custom Standards.

   The Custom Dimensions dialog box opens.

2. Select the custom standard to delete, and click **Delete**.

   The Replace Dimension dialog box opens.

3. Select a new dimension standard to replace the standard being deleted, and then click **OK**.

   Any existing dimensions that used the deleted standard are changed to the replacement standard.

Using Custom Dimension Standards

Renaming a Custom Dimension Standard

To rename a custom dimension standard:

1. Select **File > Document Settings > Document Preferences**. From the Document Preferences dialog box, click the Dimensions tab, and then click **Custom**.

   Alternatively, with any one of the dimension tools in the Dims/Notes tool set selected, open the **Dim Std** list on the Tool bar and select Custom Standards. Alternatively, with an existing dimension selected, open the **Dim Std** list on the Object Info palette and select Custom Standards.

   The Custom Dimensions dialog box opens.

2. Select the custom standard to rename, and click **Rename**.

   The Assign Name dialog box opens.

3. Enter a new name for this dimension standard and click **OK**.

Using Custom Dimension Standards

Importing a Custom Dimension Standard

To import a custom dimension standard:

1. Select **File > Document Settings > Document Preferences**. From the Document Preferences dialog box, click the Dimensions tab, and then click **Custom**.

   Alternatively, with any one of the dimension tools in the Dims/Notes tool set selected, open the **Dim Std** list on the Tool bar and select Custom Standards. Alternatively, with an existing dimension selected, open the **Dim Std** list on the Object Info palette and select Custom Standards.

   The Custom Dimensions dialog box opens.

2. Click **Import**.

   The standard Open dialog box displays.

3. Select the file from which the dimension standard will be imported and click **Open**.

   The Select Item dialog box opens.

4. Select the dimension standard to import and click **OK**.

   The imported dimension displays in the Custom Dimensions dialog box for the current drawing file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Custom Dimension Standards

Replacing a Dimension Standard

Replace any default or custom dimension standard with another dimension standard.

To replace a dimension standard:

1. Select **File > Document Settings > Document Preferences**. From the Document Preferences dialog box, click the Dimensions tab, and then click **Custom**.

   Alternatively, with any one of the dimension tools in the Dims/Notes tool set selected, open the **Dim Std** list on the Tool bar and select Custom Standards. Alternatively, with an existing dimension selected, open the **Dim Std** list on the Object Info palette and select Custom Standards.

   The Custom Dimensions dialog box opens.

2. Click **Replace**.

   The Replace Dimension Standards dialog box opens.

3. Select the dimension standard to be replaced in the **Replace** list and select the standard to replace it with in the **with** list. Click **OK**.

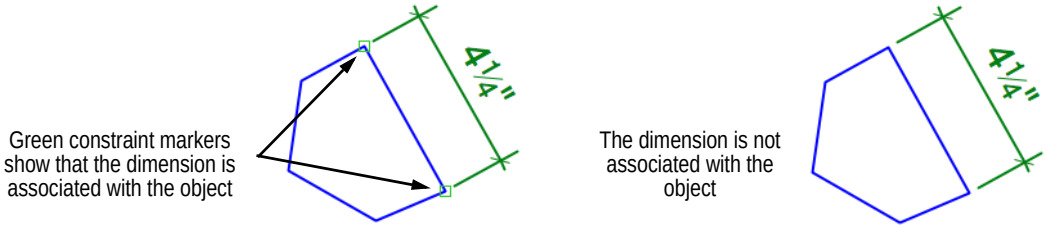
   Any existing dimensions that used the previous standard are changed to the replacement standard.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Custom Dimension Standards

Associative Dimensioning

In document preferences, the **Associate dimensions** option is selected by default. This automatically links dimensions to 2D objects, as well as walls and wall components. For linear dimensions to be associated, the dimension must be applied between two vertex points.



A dimension is linked to its associated object with coincident constraints. This means that, when a linear, radial, or diametrical dimension is associated with an object, if the dimension is resized or moved, the associated object is resized or moved also, and vice versa.

If an angular associative dimension is resized or moved, the association with the object is broken. To maintain the association, move the dimensioned object instead.

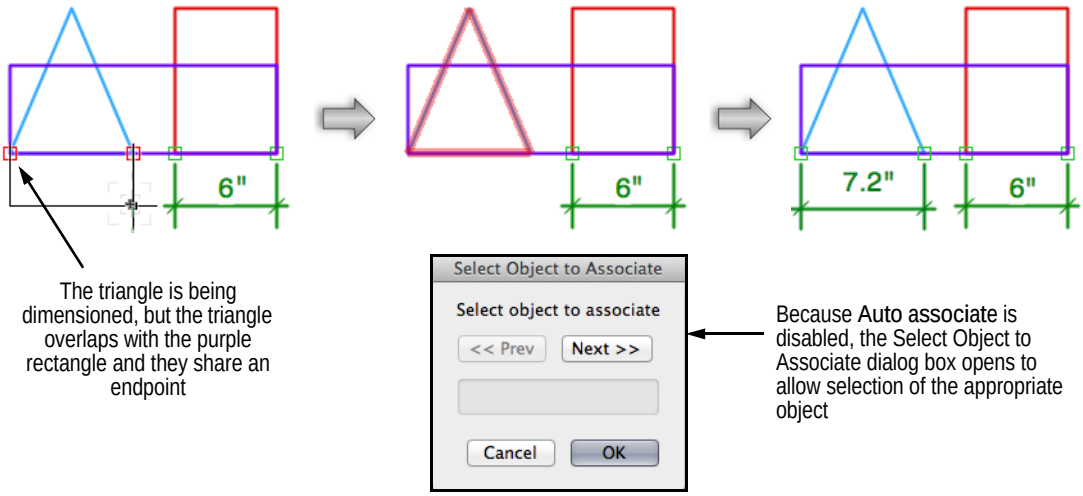
Associative dimensions can exist across layers of the same scale. To apply an associative dimension across layers, Layer Options must be set to **Show/Snap/Modify Others**. Associative dimensions can be created on viewports in annotation edit mode (see “Creating Annotations for Sheet Layer Viewports” on page 1653).

If an associative dimension is used across layers, the association is broken if the scale of one of the layers changes.

Associative dimensions can only be placed in the plane defined by the object or objects being dimensioned; when multiple objects are dimensioned, they must be co-planar to create the association. The associated dimension of an object changes planes with the object, if the object changes its plane.

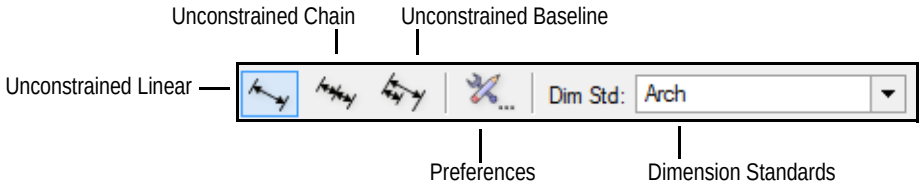
If a drawing has overlapping objects, it may be unclear which object should be associated with a dimension. When you dimension objects that have a shared endpoint, the **Auto associate** document preference controls whether the association is made automatically or manually. (See “Dimension Preferences” on page 61.)

- To always automatically associate a dimension with the top-most object, enable the **Auto associate** option.
- To manually select the object with which to associate the dimension, disable the **Auto associate** option; when you dimension an object that shares an endpoint with another object, the Select Object to Associate dialog box opens.



Unconstrained Linear Dimensioning

Use the **Unconstrained Linear Dimension** tool to draw dimension lines at any angle. This is different from the **Constrained Linear Dimension** tool, which constrains dimension lines to the X or Y axis of the plane on which the dimension is created. In addition to measuring and dimensioning objects, you can use this tool to calculate the distance between two or more points in the drawing area.



Mode	Description
Unconstrained Linear	Creates a dimension line with a single measurement
Unconstrained Chain	Creates a continuous chain of dimension lines
Unconstrained Baseline	Creates a series of connected dimension lines, starting from an initial base point
Preferences	For Unconstrained Chain mode, specifies whether dimensions are created as a single unconstrained chain object, or as individual dimension objects; also enables collision control, which automatically spaces text blocks so that they do not overlap
Dimension Standards	To change the dimension standard for new dimensions created in the document, select a standard from the list; select Custom Standards to create or manage custom dimensions (see "Using Custom Dimension Standards" on page 1187)

The dimension text can be moved as needed. See "Modifying Dimensions" on page 1210.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Unconstrained Linear Dimension
Unconstrained Chain Dimension
Unconstrained Baseline Dimension
Modifying Dimensions
Associative Dimensioning
Constrained Linear Dimensioning

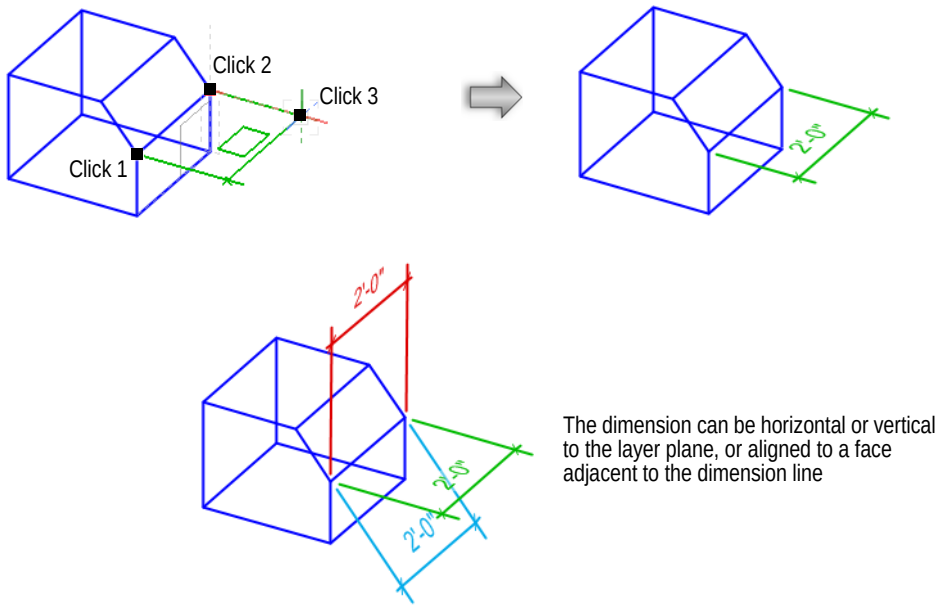
Unconstrained Linear Dimension

The Unconstrained Linear mode creates a dimension line with a single measurement.

To create an unconstrained linear dimension line with a single measurement:

1. Click the **Unconstrained Linear Dimension** tool from the appropriate tool set/palette, and select **Unconstrained Linear** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click to set the measurement start point.
3. Click to end the measurement.
4. Move the cursor away from the object to the desired offset location.

   In a 3D view, the first two clicks determine the X axis of the plane where the dimension will be placed; move the cursor to rotate the dimension plane as desired. The dimension plane can be horizontal or vertical to the layer plane, or aligned to adjacent faces of a 3D object being dimensioned.
5. Click to place the dimension line.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



The dimension can be horizontal or vertical to the layer plane, or aligned to a face adjacent to the dimension line

Unconstrained Linear Dimensioning

Unconstrained Chain Dimension

Unconstrained Chain mode creates a series of connected dimension lines, with each line segment displaying its specific measurements.

When dimensions are part of a chain object, they can all be moved at once, and their attributes and properties can be changed as a group. The properties of dimensions in a chain object also can be edited individually. When dimensions in the chain are modified, the other dimensions adjust automatically. See “Modifying Dimensions” on page 1210 for more information about editing chain objects.

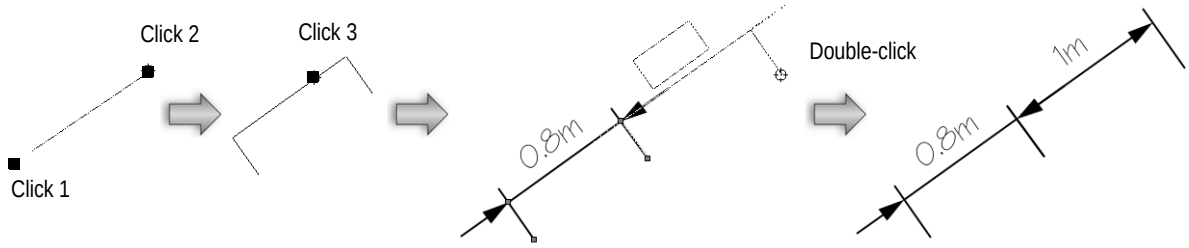
To create an unconstrained chain of connected dimension lines:

1. Click the **Unconstrained Linear Dimension** tool from the appropriate tool set/palette, and select **Unconstrained Chain** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click **Preferences** from the Tool bar.

   The Chain Dimension Preferences dialog box opens. Select whether to **Create smart chain dimension objects** or to **Create individual dimension objects adjacent to each other**. Also select whether to use collision control to automatically space text blocks so that they do not overlap. Click **OK**.
3. Click to set the measurement start point.
4. Click to end the measurement of the first segment.
5. Move the cursor away from the object to the desired offset location.

   In a 3D view, the first two clicks determine the X axis of the plane where the dimension will be placed; move the cursor to rotate the dimension plane as desired. The dimension plane can be horizontal or vertical to the layer plane, or aligned to adjacent faces of a 3D object being dimensioned.
6. Click to place the first dimension line.

7. Move the cursor to the end of the next segment and click to set its endpoint.
8. Continue setting segments.
9. Double-click to end the chain.



Unconstrained Linear Dimensioning

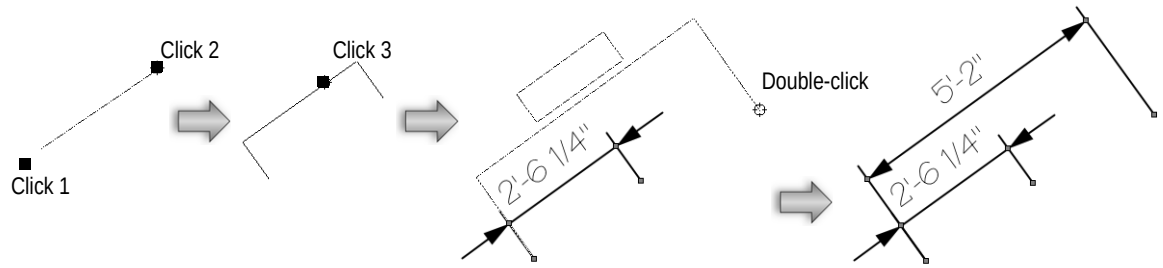
Unconstrained Baseline Dimension

Unconstrained Baseline mode creates a series of connected dimension lines, with each line segment measuring the distance from the initial starting point (the base point).

To create a series of unconstrained baseline dimension lines:

1. Click the **Unconstrained Linear Dimension** tool from the appropriate tool set/palette, and select **Unconstrained Baseline** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click to set the measurement start point.
3. Click to end the measurement of the first segment.
4. Move the cursor away from the object to the desired offset location.

   In a 3D view, the first two clicks determine the X axis of the plane where the dimension will be placed; move the cursor to rotate the dimension plane as desired. The dimension plane can be horizontal or vertical to the layer plane, or aligned to adjacent faces of a 3D object being dimensioned.
5. Click to place the dimension line.
6. Move the cursor to the end of the next segment and click to set its endpoint.
7. Continue setting segments.
8. Double-click to end the baseline.



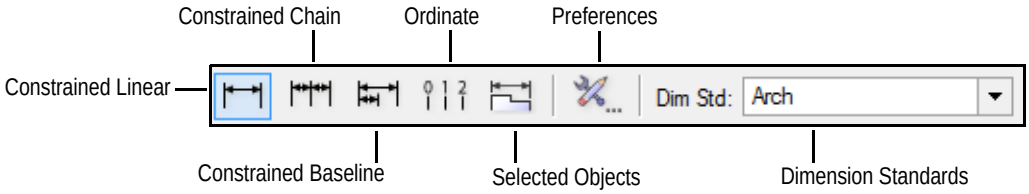
Unconstrained Linear Dimensioning

Constrained Linear Dimensioning

Use the **Constrained Linear Dimension** tool to draw dimension lines that are constrained to the X or Y axis of the plane on which the dimension is created. This is different from the **Unconstrained Linear Dimension** tool, which can draw dimension lines at any angle. In addition to measuring and dimensioning objects, you can use this tool to calculate the horizontal or vertical distance between two or more points in the drawing.

In a 2D view, the dimension line is constrained to be parallel to either the X or Y axis of the active plane. In a 3D view, the dimension line is constrained to the X axis of the plane that it is created on, but the plane can have any rotation (same as the **Unconstrained Linear Dimension** tool).

If the **Snap to Working Plane** option on the Snapping palette is enabled, the dimension line is projected to the current working plane, in both 2D and 3D views.



Mode	Description
Constrained Linear	Creates a constrained dimension line with a single measurement
Constrained Chain	Creates a continuous chain of constrained dimension lines
Constrained Baseline	Creates a series of connected constrained dimension lines, starting from an initial base point
Ordinate	Creates a series of constrained ordinate dimensions, starting from an initial base point
Selected Objects	Creates a constrained dimension line for the greatest span of a selected object or group of objects
Preferences	For Constrained Chain mode, specifies whether dimensions are created as a single chain object, or as individual dimension objects; also enables collision control, which automatically spaces text blocks so that they do not overlap
Dimension Standards	To change the dimension standard for new dimensions created in the document, select a standard from the list; select Custom Standards to create or manage custom dimensions (see “Using Custom Dimension Standards” on page 1187)

The dimension text can be moved as needed. See “Modifying Dimensions” on page 1210.

Click here for a video tip on this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Constrained Linear Dimension
Constrained Chain Dimension
Constrained Baseline Dimension
Ordinate Dimensions
Selected Object Dimensions
Modifying Dimensions
Associative Dimensioning
Unconstrained Linear Dimensioning

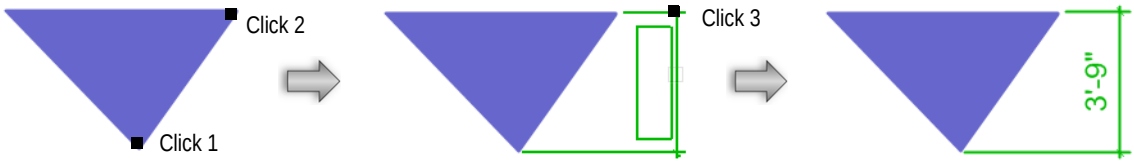
Constrained Linear Dimension

The Constrained Linear mode creates a dimension line with a single measurement.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To create a constrained linear dimension line with a single measurement:

1. Click the **Constrained Linear Dimension** tool from the appropriate tool set/palette, and select **Constrained Linear** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click to set the measurement start point.
3. Click to set the end of the measurement.
4. Move the cursor away from the object to the desired offset location.

   In a 2D view, the dimension line preview is constrained to be parallel to either the X or Y axis of the layer plane. In a 3D view, the first two clicks determine the X axis of the plane where the dimension will be placed; move the cursor to rotate the dimension plane as desired. The dimension plane can be horizontal or vertical to the layer plane, or aligned to adjacent faces of a 3D object being dimensioned.
5. Click to place the dimension line.



Constrained Linear Dimensioning

Constrained Chain Dimension

The Constrained Chain mode creates a series of constrained, connected dimension lines, with each line segment displaying its specific measurements.

When dimensions are part of a chain object, they can all be moved at once, and their attributes and properties can be changed as a group. The properties of dimensions in a chain object also can be edited individually. When dimensions in the chain are modified, the other dimensions adjust automatically. See “Modifying Dimensions” on page 1210 for more information about editing chain objects.

To create a chain of constrained connected dimension lines:

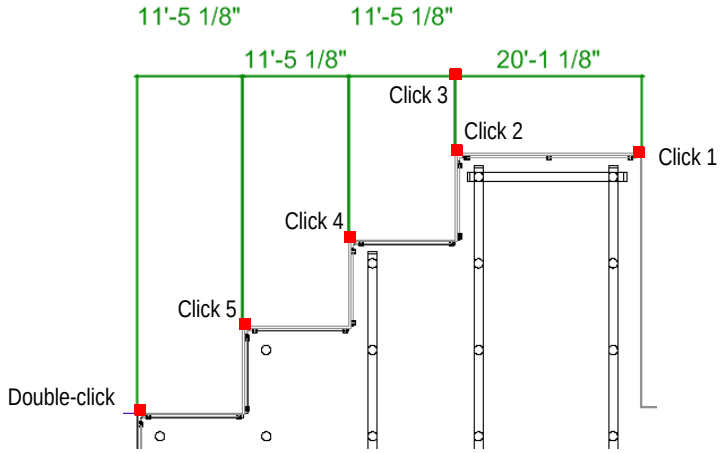
1. Click the **Constrained Linear Dimension** tool from the appropriate tool set/palette, and select **Constrained Chain** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click **Preferences** from the Tool bar.

   The Chain Dimension Preferences dialog box opens.

   Select whether to **Create smart chain dimension objects** or to **Create individual dimension objects adjacent to each other**. Also select whether to use collision control to automatically space text blocks so that they do not overlap. Click **OK**.
3. Click to set the measurement start point.
4. Click to set the end of the first measurement.
5. Move the cursor away from the object to the desired offset location.

In a 2D view, the dimension line preview is constrained to be parallel to either the X or Y axis of the layer plane. In a 3D view, the first two clicks determine the X axis of the plane where the dimension will be placed; move the cursor to rotate the dimension plane as desired. The dimension plane can be horizontal or vertical to the layer plane, or aligned to adjacent faces of a 3D object being dimensioned.

6. Click to place the first dimension line.
7. Move the cursor to the end of the next segment and click to set its endpoint.
8. Continue setting segments.
9. Double-click to end the chain.



Constrained chain dimension with collision control preference enabled

Constrained Linear Dimensioning

Constrained Baseline Dimension

Constrained Baseline modes create a series of constrained, connected dimension lines, with each line segment measuring the distance from the initial starting point (the base point).

To create a series of constrained baseline dimension lines:

1. Click the **Constrained Linear Dimension** tool from the appropriate tool set/palette, and select **Constrained Baseline** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click to set the measurement start point.
3. Click to set the end of the first measurement.
4. Move the cursor away from the object to the desired offset location.

   In a 2D view, the dimension line preview is constrained to be parallel to either the X or Y axis of the layer plane. In a 3D view, the first two clicks determine the X axis of the plane where the dimension will be placed; move the cursor to rotate the dimension plane as desired. The dimension plane can be horizontal or vertical to the layer plane, or aligned to adjacent faces of a 3D object being dimensioned.
5. Click to place the first dimension line.
6. Move the cursor to the end of the next segment and click to set its endpoint.
7. Continue setting segments.

8. Double-click to end the baseline.

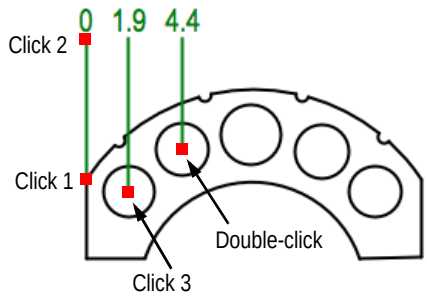
Constrained Linear Dimensioning

Ordinate Dimensions

The Ordinate mode measures and dimensions a series of either horizontal or vertical distances from one fixed point. This mode of the **Constrained Linear Dimension** tool always creates dimensions on the active plane.

To draw constrained ordinate dimensions:

1. Click the **Constrained Linear Dimension** tool from the appropriate tool set/palette, and select **Ordinate** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click to set the measurement start point; the first segment is always considered the fixed point and is labeled 0.
3. Click to set the length of the first segment.
4. Move the cursor horizontally or vertically to the first location to be measured from the initial segment; click to set the segment.
5. Continue creating segments.
6. Double-click to complete the dimension.



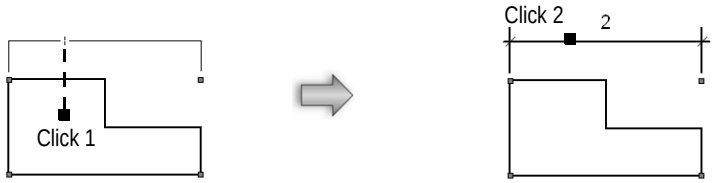
Constrained Linear Dimensioning

Selected Object Dimensions

The Selected Objects mode measures and dimensions the horizontal or vertical span of any 2D or 3D object, or the edge-to-edge span of several objects or a group of objects. In all cases, the tool measures the greatest span. Horizontal dimension lines can be drawn inside, above, or below an object or group of objects. Vertical dimension lines can be drawn inside, to the right, or to the left of an object or group of objects. This mode of the **Constrained Linear Dimension** tool always creates dimensions on the active plane.

1. Select the object or objects to dimension.
2. Click the **Constrained Linear Dimension** tool from the appropriate tool set/palette, and select **Selected Objects** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
3. Click near the object or group of objects to be dimensioned.
4. Move the cursor in the desired direction where the dimension should be created. A preview dimension displays, constrained in either the horizontal or vertical direction.

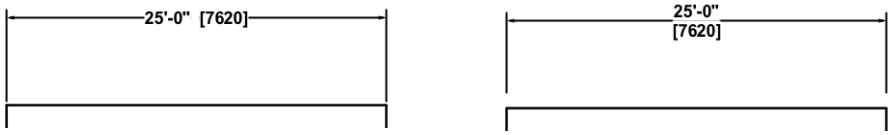
5. Click to set the position of the dimension line.



Constrained Linear Dimensioning

Dual Dimensioning

Dual dimensioning displays two sets of values, such as inches and millimeters, within a single dimension. These values have independent unit settings and attributes. Dual dimensions can be displayed side by side or stacked.



Side-by-side dual dimension Stacked dual dimension

Control the units for each dimension through **File > Document Settings > Units**. The primary dimension uses the **Units** set on the General Display and Dimensions tab of the Units dialog box. The settings on the Dual Dimensions tab provide full control of the unit of measurement, unit marks, rounding, and formatting used for the secondary dimension.

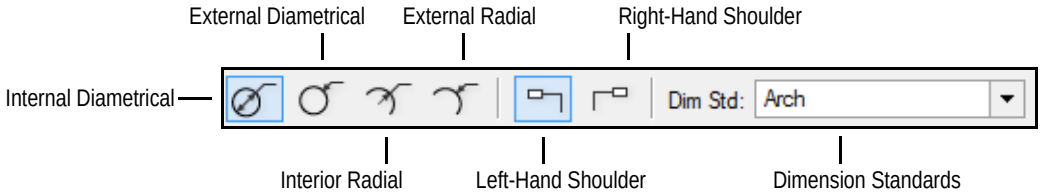
Once a dual dimension is placed on the drawing, adjust the individual attributes for both the primary and secondary dimension from the Object Info palette. The **Dual View** and **Prim/Sec** attributes apply to dual dimensions.

Modifying Dimensions
Units

Radial Dimensioning

The **Radial Dimension** tool measures and marks either radius or diameter dimensions for arcs and circles. This tool does not work on ovals or objects with rounded corners.

The **Radial Dimension** tool can place the dimension lines and measurements either inside or outside of the circle/arc. This tool always creates the dimension on the same plane with the arc or circle that is being dimensioned.



Mode	Description
Internal Diametrical	Measures and dimensions the diameter of a circle or arc, placing the dimension inside the object
External Diametrical	Measures and dimensions the diameter of a circle or arc, placing the dimension outside the object

Mode	Description
Interior Radial	Measures and dimensions the radius of a circle or arc, placing the dimension inside the object
External Radial	Measures and dimensions the radius of a circle or arc, placing the dimension outside the object
Left-Hand Shoulder	Places an exterior measurement with the dimension text to the left of the dimension leader shoulder
Right-Hand Shoulder	Places an exterior measurement with the dimension text to the right of the dimension leader shoulder
Dimension Standards	To change the dimension standard for new dimensions created in the document, select a standard from the list; select Custom Standards to create or manage custom dimensions (see “Using Custom Dimension Standards” on page 1187)

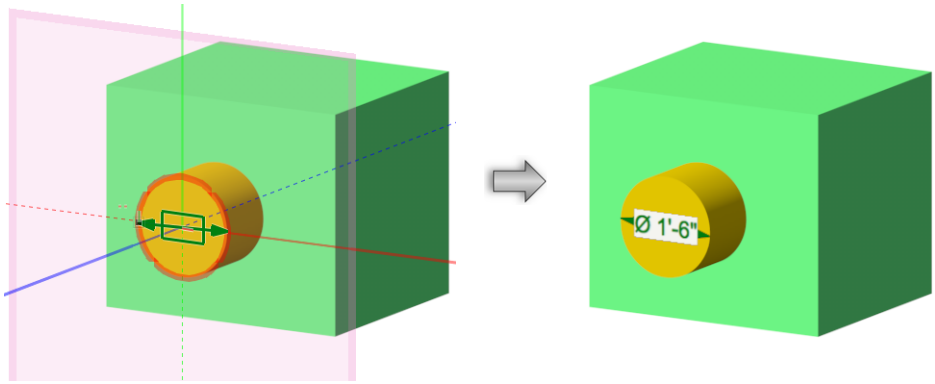
Diametrical Dimensioning Inside Circle/Arc
Diametrical Dimensioning Outside Circle/Arc
Radial Dimensioning Inside Circle/Arc
Radial Dimensioning Outside Circle/Arc
Modifying Dimensions
Associative Dimensioning

Diametrical Dimensioning Inside Circle/Arc

The Internal Diametrical mode measures and dimensions the diameter of a circle or arc, and places the dimension inside the object.

To dimension the diameter of a circle or arc and place the dimension inside the object:

1. Click the **Radial Dimension** tool from the appropriate tool set/palette, and select **Internal Diametrical** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click on or within the arc or circle to dimension.
3. Move the cursor to preview the dimension line location.
4. Click to place the dimension.



Radial Dimensioning

Modifying Dimensions

Diametrical Dimensioning Outside Circle/Arc

The External Diametrical mode measures and dimensions the diameter of a circle or arc, and places the dimension outside the object.

To dimension the diameter of a circle or arc and place the dimension outside the object:

1. Click the **Radial Dimension** tool from the appropriate tools set/palette, and select **External Diametrical** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Specify the side of the dimension line to place the measurement.

   To place it to the right of the line, click **Right-Hand Shoulder** mode button. To place it to the left of the line, click the **Left-Hand Shoulder** mode button.
3. Click on or within the arc or circle to dimension.
4. Move the cursor to preview the dimension line location.
5. Click to place the dimension.

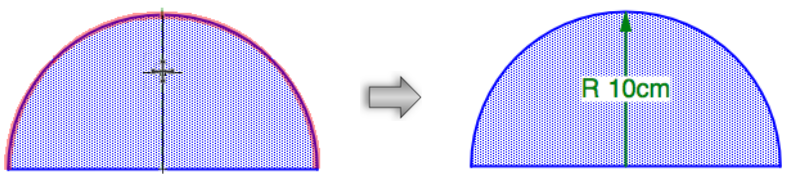
Radial Dimensioning
Modifying Dimensions

Radial Dimensioning Inside Circle/Arc

The Interior Radial mode measures and dimensions the radius of a circle or arc, and places the dimension inside the object.

To dimension the radius of a circle or arc and place the dimension inside the object:

1. Click the **Radial Dimension** tool from the appropriate tool set/palette, and select **Interior Radial** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click on or within the arc or circle to dimension.
3. Move the cursor to preview the dimension line location.
4. Click to place the dimension.



Radial Dimensioning
Modifying Dimensions

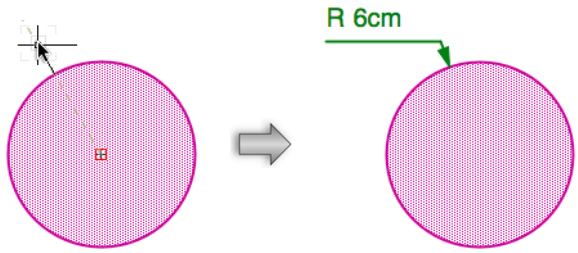
Radial Dimensioning Outside Circle/Arc

The External Radial mode measures and dimensions the radius of a circle or arc, and places the dimension outside the object.

To dimension the radius of a circle or arc and place the dimension outside the object:

1. Click the **Radial Dimension** tool from the appropriate tool set/palette, and select **External Radial** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Specify the side of the dimension line to place the measurement.

   To place it to the right of the line, click the **Right-Hand Shoulder** mode button. To place it to the left of the line, click the **Left-Hand Shoulder** mode button.
3. Click on or within the arc or circle to dimension.
4. Move the cursor to preview the dimension line location.
5. Click to place the dimension.



Radial Dimensioning
Modifying Dimensions

Marking Object Centers

The **Center Mark** tool divides an object or object face into quarters, marking the exact center of the object. This tool works for circles, ovals, rectangles, rounded rectangles, and faces of 3D objects that were created by a circular edge. In addition, it can place center marks on the corners of a rounded rectangle.

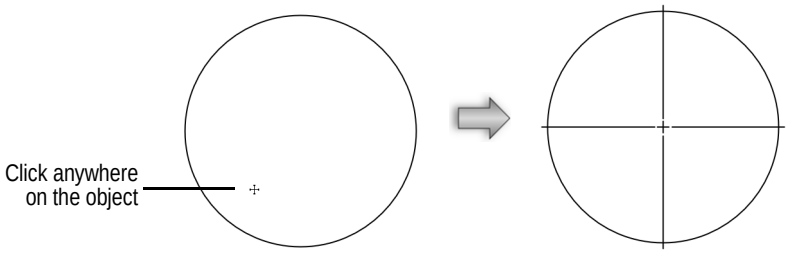
When the **Center Mark** tool is active, you can change the dimension standard for new dimensions created in the document from the Tool bar. To do so, select a standard from the **Dim Std** list, or select Custom Standards to create or manage custom dimensions (see “Using Custom Dimension Standards” on page 1187).

Marking the Center of an Object

To mark the center of an object:

1. Click the **Center Mark** tool from the Dims/Notes tool set.
2. Click on the object to mark.

   The center is marked by the intersection of two lines.

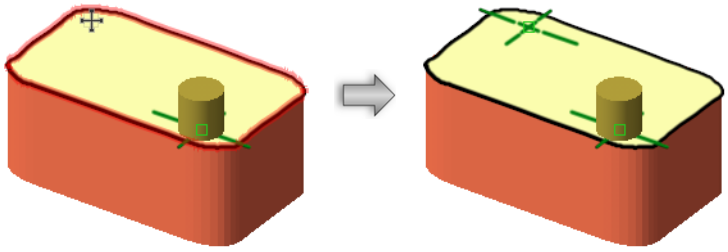


Marking the Center of a Rounded Rectangle Corner

To place a center mark in the corners of a rounded rectangle:

1. Click the **Center Mark** tool from the Dims/Notes tool set.
2. While pressing Option (Mac) or Alt (Windows), move the cursor over the rounded rectangle corner to mark.
3. Click to place the center mark.

   The center is marked by the intersection of two lines.

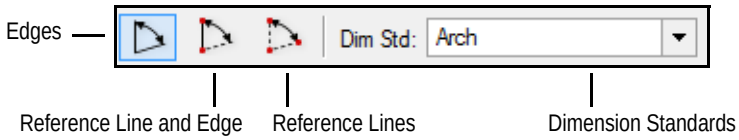


Modifying Dimensions

Angular Dimensioning

The **Angular Dimension** tool measures and dimensions angles. Dimension the angle between two object edges, between two sides of a single object, between a single object and a reference line, or between two reference lines.

This tool works with all objects with linear or planar sides, including rounded and rotated rectangles, lines, polylines, polygons, and solids. It does not, however, work with circles, ovals, or arcs. In addition, it cannot dimension between parallel lines or lines on different planes.



If the **Snap to Working Plane** option on the Snapping palette is enabled, the face of the object being dimensioned must be parallel to the working plane.

Mode	Description
Edges	Dimensions the angle between two object edges or faces
Reference Line and Edge	Dimensions the angle between an object and a reference line
Reference Lines	Dimensions the angle between two reference lines

Mode	Description
Dimension Standards	To change the dimension standard for new dimensions created in the document, select a standard from the list; select Custom Standards to create or manage custom dimensions (see “Using Custom Dimension Standards” on page 1187)

Angle Between Two Object Edges or Faces
Angle Between One Reference Line and an Object
Angle Between Two Reference Lines
Modifying Dimensions
Associative Dimensioning

Angle Between Two Object Edges or Faces

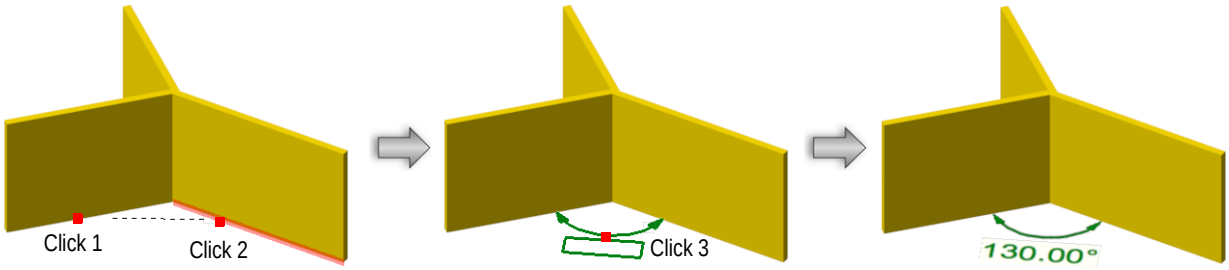
To dimension the angle between the edges or faces of two objects:

1. Click the **Angular Dimension** tool from the appropriate tool set/palette, and select the **Edges** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click the edge or face of the first object.

   A preview line displays.
3. Click the edge or face of the second object.

   An angular dimension preview displays.

   To select a different angle—for example, spanning the opposite direction—move the cursor until the preview displays the desired angle.
4. Click a third time to define the radius of the dimension.



Angular Dimensioning
Modifying Dimensions

Angle Between One Reference Line and an Object

To dimension an angle between one reference line and an object:

1. Click the **Angular Dimension** tool from the appropriate tool set/palette, and select the **Reference Line and Edge** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette

2. Click to set the start of the reference line.

   A reference line preview displays.

3. Click again to set the end of the reference line.

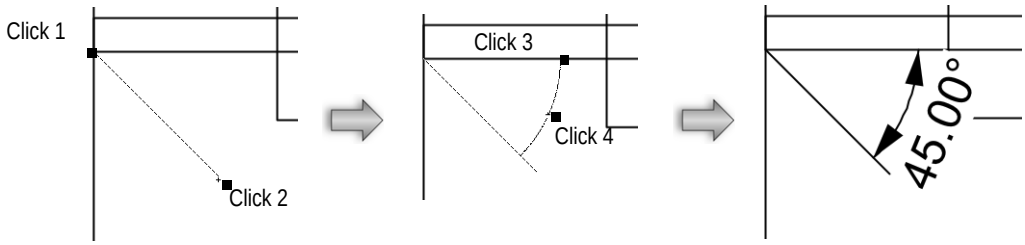
   The cursor changes to a pointing hand.

4. Click on the edge of the object (and, therefore, angle) to dimension.

   An angular dimension preview displays.

   To select a different angle—for example, spanning the opposite direction—move the cursor until the preview displays the desired angle.

5. When the preview displays the desired angle, click again to draw the reference line and to define the angle of the dimension.



Angular Dimensioning
Modifying Dimensions

Angle Between Two Reference Lines

To dimension an angle between two references lines:

1. Click the **Angular Dimension** tool from the appropriate tool set/palette, and select the **Reference Lines** mode.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click to set the start of the first reference line.

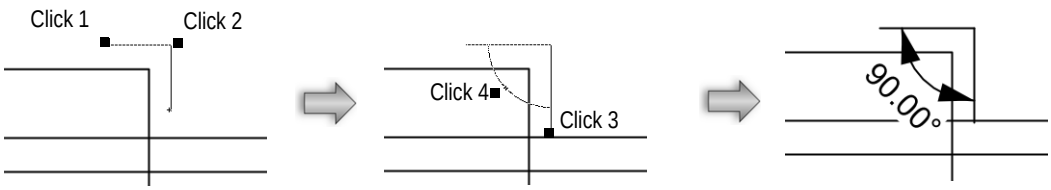
   A reference preview line displays.

3. Click again to set the end of the first reference line.
4. A second reference line preview displays.
5. Click a third time to set the end of the second reference line.

   An angular dimension preview displays.

   To select a different angle—for example, spanning the opposite direction—move the cursor until the preview displays the desired angle.

6. When the preview displays the desired angle, click again to define the angle of the dimension.

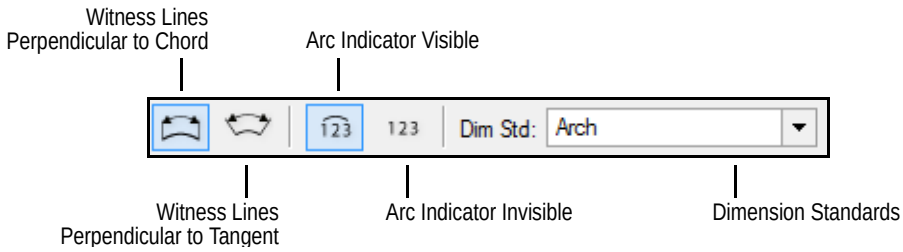


Angular Dimensioning
Modifying Dimensions

Arc Length Dimensioning

The **Arc Length Dimension** tool measures and dimensions the length of an arc anywhere along its circumference. The dimension can be shown with its witness lines perpendicular to a chord on the arc, or perpendicular to a line tangent to the arc. There are also modes to hide or show a graphic of an arc over the dimension measurement.

This tool works with arcs, polylines that include arc segments, arc-based 3D objects, and round walls. It always creates the dimension on the same plane with the arc that is being dimensioned.

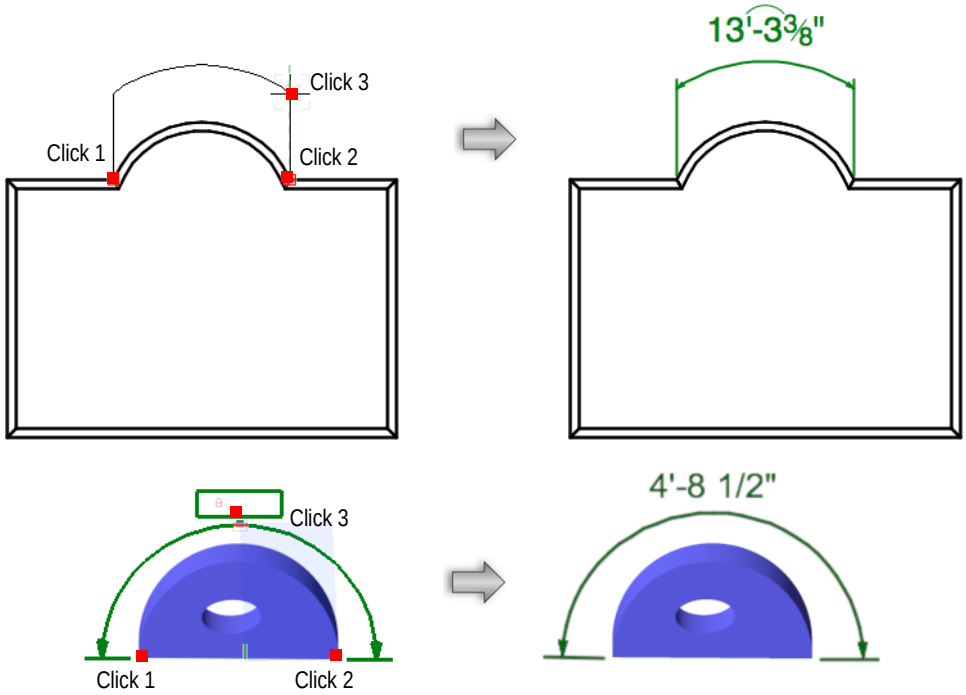


Mode	Description
Witness Lines Perpendicular to Chord	Dimensions the arc with witness lines that would be perpendicular to a chord drawn across the arc
Witness Lines Perpendicular to Tangent	Dimensions the arc with witness lines that would be perpendicular to a line drawn tangent to the arc
Arc Indicator Visible	Displays an arc graphic over the dimension measurement
Arc Indicator Invisible	Uses a dimension measurement with no arc graphic
Dimension Standards	To change the dimension standard for new dimensions created in the document, select a standard from the list; select Custom Standards to create or manage custom dimensions (see “Using Custom Dimension Standards” on page 1187)

 To create an arc length dimension:

1. Click the **Arc Length Dimension** tool from the appropriate tool set/palette.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Select the desired mode for the display of the witness lines and whether to use the arc indicator over the dimension measurement.
3. Click to set the measurement start point.
4. Click to set the end of the measurement.
5. Move the cursor away from the object.

   This specifies how far the dimension line is offset from the measured object.
6. Click to place the dimension line.

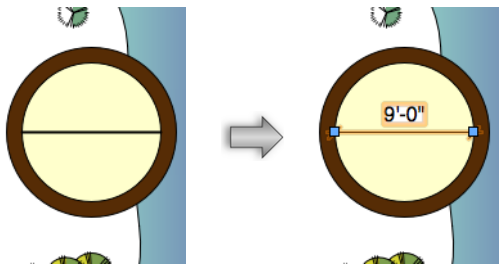


Modifying Dimensions
Associative Dimensioning

Converting Objects to Dimensions

To convert a line, double line, or arc to a dimension:

1. Select the object to convert.
2. Select **Modify > Convert > Convert Object to Dimension**.
3. The object is converted to a dimension. Lines change to linear dimensions, and arcs change to angular dimensions.



Modifying Dimensions

Dimensioning Exterior Walls

The exterior walls of a building can be automatically dimensioned with the **Dimension Exterior Walls** command. The command can:

- dimension exterior walls by outside edge or center line;
- associate dimensions with walls;
- dimension only visible wall components; and

- dimension all windows and doors in the exterior walls by the center line or the edge of the openings.

  For window objects, the rough opening is calculated as the unit size plus two times the shim gap. For door objects, the rough opening is calculated as the leaf size plus two times the jamb width plus two times the shim gap.

  The exterior wall dimensioner uses the dimension standard specified on the Dimensions tab of the document preferences (**File > Document Settings > Document Preferences**) and the unit selection and precision settings in **File > Document Settings > Units**.

To generate exterior wall dimensions:

1. In a file with exterior walls, determine whether you wish to dimension to the structural or non-structural components of the wall. To dimension to the structural components, such as framing components, make the classes of the non-structural components invisible.

   The wall styles available in the Vectorworks Architect product contain pre-classed components, making it easy to show and hide the desired portions of the wall.

2. Select the **Dimension Exterior Walls** command from the appropriate menu:

   - Architect workspace: **AEC > Dimension Exterior Walls**
   - Landmark workspace: **Landmark > Architectural > Dimension Exterior Walls**
   - Spotlight workspace: **Spotlight > Architectural > Dimension Exterior Walls**

   The Dimension Exterior Walls dialog box opens. Specify how to dimension the walls.

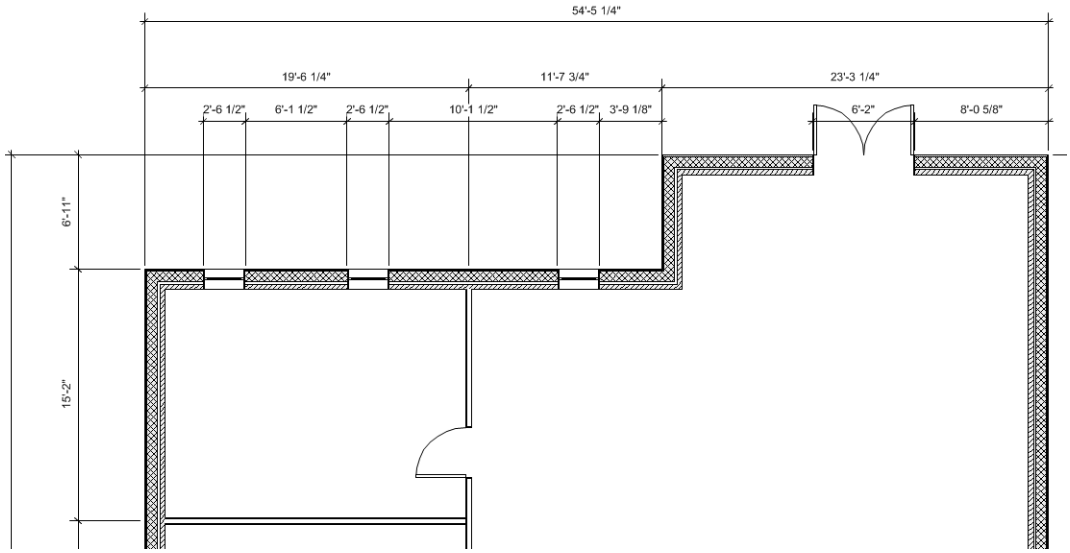
Click to show/hide the parameters.

Parameter	Description
Source Layer	Specifies the location of the walls to be dimensioned
Options	Specify the dimension methods
Dimension T-joins in exterior walls	Interior walls which intersect with the exterior walls are dimensioned
Make dimensions associative	Associates dimensions with the wall or wall component; dimensions move and update with the wall or wall component
Dimension to Wall	Specifies whether to dimension to the wall extents or center lines
Dimension to	Specifies whether to dimension to the edge of the openings or to the center of the openings Rough openings are automatically defined in door and window objects. To add a rough opening to a symbol inserted in a wall, draw an invisible polygon (line weight 0, no fill) with three vertices to the extents of the rough opening.
Text Options	Specifies dimension text properties
Rotation	Controls the orientation of the dimension text • Horiz/Vert places text horizontally for horizontal dimensions and vertically for vertical dimensions • Horizontal keeps text always horizontal • Aligned places text according to the dimension angle
Location	Specifies whether the text should be centered between the witness lines if space permits, or always placed outside the witness lines
If forced outside, distance	Indicates the offset for text forced outside the witness lines

Parameter	Description
Above Dim Line	Select to always place text above the dimension line
Offsets from Building	Specifies the distance of the dimensions from the building, walls, doors and windows
Small Dimensions	Specifies the limit for drawing small dimensions, which can help detect inaccuracies in the drawing; dimensions less than the value specified will be considered “small.” Select **Draw in red** to detect problems such as minute misalignments in layout; to overlook any inaccuracies, select **Do not draw**.

3. Click **OK**.

   The exterior wall dimensions are drawn automatically.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Wall Styles
Dimensions

Modifying Dimensions

Modify dimensions by adjusting parameters in the Object Info palette or the Properties dialog box. Use commands on the object context menu to format dimension text, to delete and edit individual dimensions within a chain, to add a dimension to a single or chain linear dimension, and to disassociate dimensions from the objects to which they are linked. You can also make many common adjustments by manipulating dimensions directly with the **Selection** tool; see “Editing Dimensions with the Mouse” on page 1214.

> To change adjoining dimensions into a single chain dimension object, select them, and then select **Modify > Compose**. Similarly, to change a chain object into multiple dimensions, select the chain, and then select **Modify > Decompose**.

The association of any dimension with an object can be removed; however, special considerations apply when an object also has parametric constraints.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modification	Method	Description
Disassociate a dimension from an object that has parametric constraints	Select the object, then right-click (Windows) or Ctrl-click (Mac) and select **Disassociate** from the context menu	"Associative Dimensioning" on page 1191
Disassociate specific dimensions from an object that has parametric constraints, without affecting the parametric constraints	Select the object, select **Modify > Edit Constraints**, and then select the associated dimensions to remove	"Editing Parametric Constraints" on page 1237
Disassociate all of the dimensions from an object that has parametric constraints, as well as remove all of the parametric constraints	Select the object, then right-click (Windows) or Ctrl-click (Mac) and select **Remove Constraints** from the context menu	"Deleting Parametric Constraints" on page 1237

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

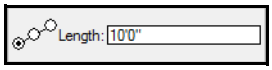
Editing Dimension Properties

Edit the properties of one or more selected dimensions in the Object Info palette. Alternatively, right-click (Windows) or Ctrl-click (Mac) on the dimension, and select **Properties** from the context menu.

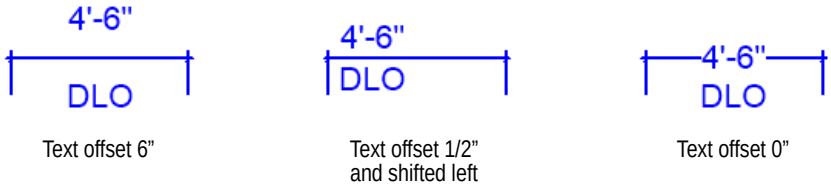
For chain dimension objects, the edits are applied to all dimensions within the chain. To edit the properties of individual dimensions within a chain, right-click (Windows) or Ctrl-click (Mac) on the dimension, and select **Edit Dimension** from the context menu instead.

Different parameters are available depending on what type of dimension is being edited.

Click to show/hide the parameters.

Parameter	Description
Length	For linear and baseline dimensions, and for individual dimensions within a chain, this sets the length of the dimension. For associative dimensions, this also changes the length of the associated object. Use the segment position selector to the left of the field label to specify which segment of the dimension (either endpoint, or the center point) will remain fixed when the dimension is resized. This fixed point is a global setting for all editing of linear and baseline dimension lengths. 
Radius	For radial and diametrical dimensions, this sets the length of the dimension (without moving the center point of the dimension). For associative dimensions, this also changes the radius of the associated object.
Dim Std	Selects the dimension standard type; this selection determines which fields display in the Object Info palette or Properties dialog box.
Dim Off	Sets the distance that the dimension line is offset from the dimensioned object
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Arrows Inside	Sets whether arrows display inside witness lines or are flipped outside
Interior Arc	Sets the angular dimension inside the witness lines; deselect to move the dimension outside of the witness lines at the opposite angle
Leader to Left	Switches the radial dimension leader from the right to the left side of the object
Witness Lines	
Visible	For linear and arc dimensions, this sets whether the witness line displays on both ends of the dimension line, only the start, only the end, or not at all. For chain dimension objects, this sets whether all or none of the witness lines display.
Wit Ang	For arc dimensions, sets whether the witness lines are perpendicular to a chord on the arc or to a tangent to the arc
Override	For linear and arc dimensions, this displays additional fields for you to enter override values for the length of one or both witness lines, or for the distance that one or both witness lines are offset from the dimensioned object. For chain dimension objects, by default there is only a single override value for the length or offset distance for all witness lines. To set an override for a particular dimension, right-click (Windows) or Ctrl-click (Mac) on the dimension, and select **Edit Dimension** from the context menu.
Text	
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Text Off	Sets the distance that the dimension text is offset from the dimension line. For non-aligned text, the offset is measured from the closest (left or right) edge of the text box. A value of zero places the text within the line. Depending on the type of dimension, and the **Text Rot** setting, the **Text Off** field may or may not be applicable. • Linear dimensions: text can be offset for any **Text Rot** setting • Arc and angle dimensions: only aligned text can be offset • Radial dimensions: aligned text can be offset whether inside or outside the circle; non-aligned text can only be offset when it is outside the circle
Text Rot	Sets the orientation of the dimension text to the dimension line, even when the dimension text is moved. • Horizontal: keeps the text horizontal • Aligned: keeps the text aligned parallel to the dimension line • Horiz/Vert: keeps the text vertical for a vertical linear dimension, and horizontal in all other cases
Auto Position Text	Automatically aligns the dimension text to the center of the dimension line; becomes deselected automatically when you drag the text to a new location
Flip Text	Mirrors the dimension text to the opposite side of the dimension line
Box Text	Places a box around the dimension value

Parameter	Description
Arc Indicator	Displays an arc graphic over the dimension value
Leader Line	For linear, chain, and arc dimensions, draws a leader line from the dimension value to the dimension line. If desired, use the Attributes palette to add an end marker to the leader line (see “Marker Attributes” on page 1102); the end marker defaults to the style specified for the current dimension standard, but it can be changed.
Dual View	Selects which dimensions to display, when a dual-dimension standard is selected. This parameter is not available for chain dimension objects.
Prim/Sec	Toggles between settings for primary and secondary dimensions, when a dual-dimension standard is selected. This parameter is not available for chain dimension objects.
Prec	Sets the dimension precision with up to ten digits of accuracy
Show Dim Value	Shows or hides the dimension value; **Note** text is not affected by this setting
Leader	Enter text to display before the dimension value
Trailer	Enter text to display after the dimension value
Note	Enter note text to display on the opposite side of the dimension line from the dimension value. (If the dimension value displays in the middle of the line, the note displays on the next paragraph.) If the dimension value is moved, the note moves with it, and maintains the same offset distance from the line, in the opposite direction. The note text has the same font, size, and style as the dimension value. 
Tol	When a single-dimension standard is selected, sets whether a dimension displays a single, double, limited, or no tolerance value
Top/Bottom/ Display as Typed	When a tolerance display is selected, sets the tolerance values and how they display

Modifying Dimensions

Editing Dimensions with the Context Menu

The editing commands available from the context menu depend on the dimension type. For chain dimensions, the commands also depend on where you click the dimension.

Command	Dimension Type	Description
Add Dimension	Linear and chain	Right-click (Windows) or Ctrl-click (Mac) anywhere on the dimension object, and select the command. Click with the bull’s-eye cursor to set the endpoint of the new witness line. The other dimensions in the chain adjust automatically. If **Create smart chain dimension objects** is enabled in the chain dimension preferences, a single linear dimension automatically becomes a chain when you add a dimension to it.

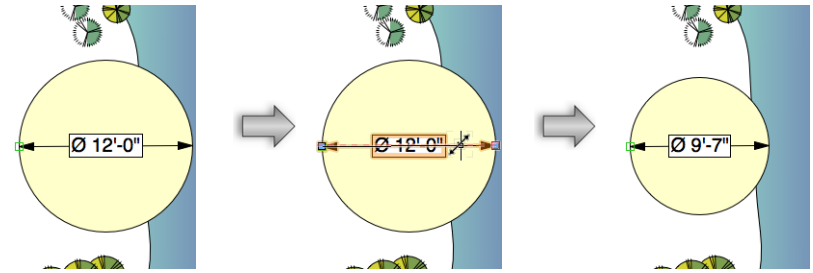
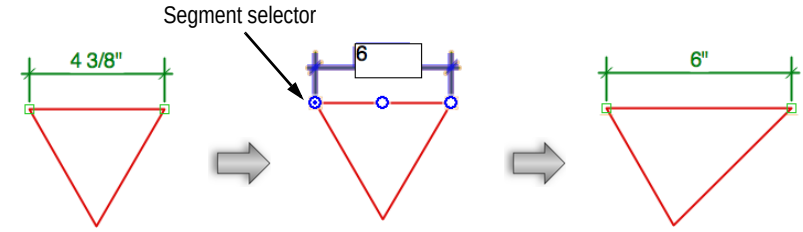
Command	Dimension Type	Description
Delete Dimension	Chain	Right-click (Windows) or Ctrl-click (Mac) on the dimension line of the dimension to be deleted, and select the command. The remaining dimensions in the chain adjust automatically.
Delete Segment/ Witness Line	Chain	Right-click (Windows) or Ctrl-click (Mac) on the segment/witness line to be deleted, and select the command. The remaining dimensions in the chain adjust automatically.
Edit Dimension	Chain	Right-click (Windows) or Ctrl-click (Mac) on the dimension to be edited, and select the command. Adjust the settings in the Object Properties dialog box and click **OK** (see "Editing Dimension Properties" on page 1211). To apply properties edits to all dimensions in the chain, use the **Properties** command or the Object Info palette instead.
Format Text	All	Right-click (Windows) or Ctrl-click (Mac) on the dimension, and select the command. Adjust the settings in the Format Text dialog box, and click **OK**.
Disassociate	Linear, radial, and diametrical	Right-click (Windows) or Ctrl-click (Mac) on the dimension, and select the command.
Properties	All	Right-click (Windows) or Ctrl-click (Mac) on the dimension, and select the command. Adjust the settings in the Object Properties dialog box and click **OK** (see "Editing Dimension Properties" on page 1211). To apply properties a specific dimension in a chain, use the **Edit Dimension** command instead.

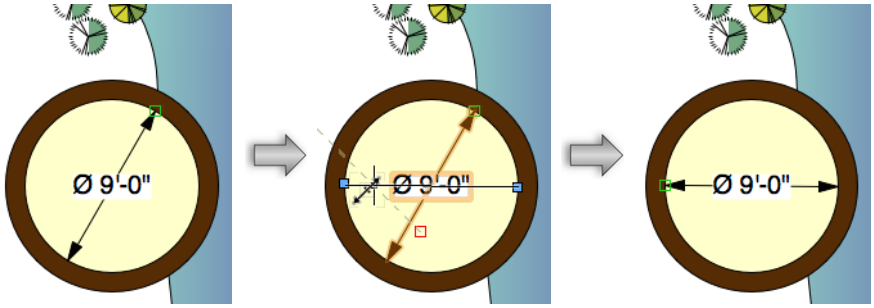
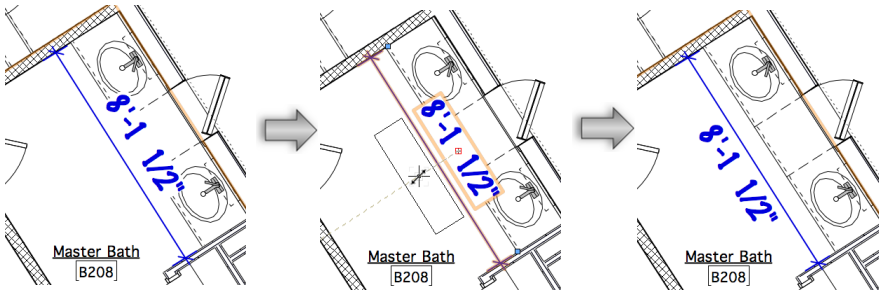
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

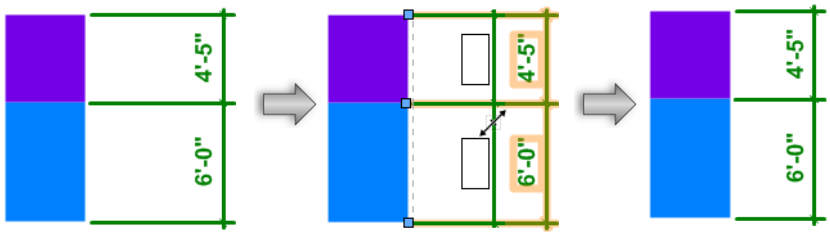
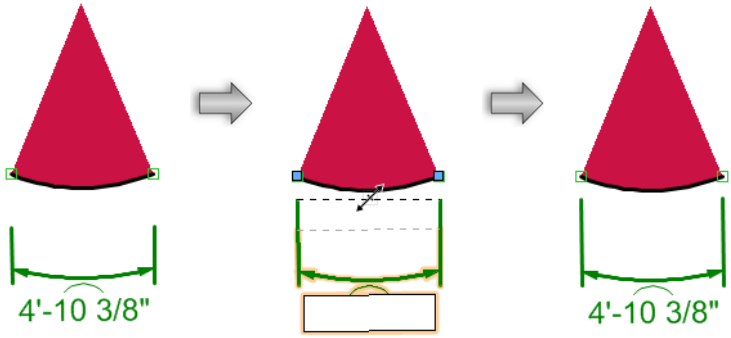
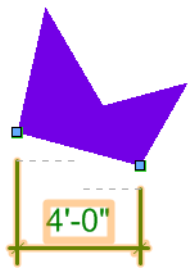
Modifying Dimensions

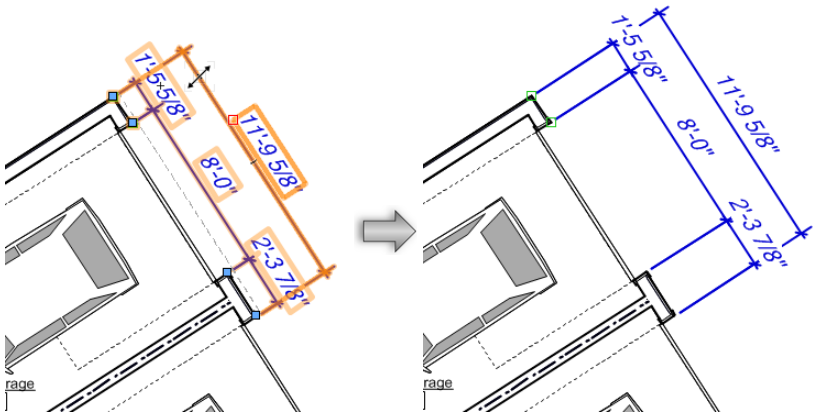
Editing Dimensions with the Mouse

All dimension parameters are available in the Object Info palette and the Properties dialog box. Dimensions also have several controls that allow you to make common adjustments quickly and directly with the mouse.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Action	Description
Change the length of a dimension graphically	• For linear dimensions, click on the endpoint of a witness line with the **Selection** tool and drag it parallel to the dimension line; in chain dimension objects, the size of any adjoining dimension is adjusted automatically. • For angular, arc length, radial, or diametrical dimensions, click on the endpoint of a witness line with the **Selection** tool and drag it in the desired direction. For associative dimensions, a change to the length of a radial or diametrical dimension also changes the size of the associated object. To enter a specific value, use the **Length** field on the Object Info palette.  The dimension has a constraint marker, so it is associative Click the right endpoint of the dimension and drag it to the left The circle's diameter is reduced from the right; the left endpoint remains fixed
Enter the length of a dimension on the drawing (linear and chain dimensions)	Double-click anywhere on the dimension to enter a text editing mode. Enter the desired length, and then use the segment position selector to specify which segment will remain fixed when the dimension is resized. Alternatively, select the **Text** tool, and then click the dimension text to enter the text editing mode. For associative dimensions, a change to the length of a linear dimension also changes the size of the associated object.  The dimension has constraint markers, so it is associative Double-click the dimension to edit it; enter the new length, and click the left segment selector The triangle's top side is expanded from the right; the dimension's left endpoint remains fixed

Action	Description
Change the rotation of a radial or diametrical dimension	Click anywhere on the witness line with the **Selection** tool and drag it in the desired direction 
Change the position of the dimension text	Click on the text with the **Selection** tool and drag it to the desired location. The text for an angular dimension can only be moved along the curve of the dimension itself; the text for other dimensions can be moved in any direction. For linear and arc length dimensions, press and hold the Shift key while dragging the text to maintain the same offset distance from the dimension line. To enter a specific value for the text offset from the dimension line, use the **Text Off** field on the Object Info palette. For linear, arc length, and chain dimensions, you can add a leader line from the text to the dimension line if necessary.  The dimension obscures part of the drawing Click the dimension text and drag it to the other side of the line
Move a dimension	With the **Selection** tool, click on a witness line (but not on an endpoint) or on one of the two control points at each end of a dimension line, and drag the entire dimension to a new location. For associative dimensions, this also moves the associated object.

Action	Description
Change the offset of the dimension line from the dimensioned object (linear, chain, and arc dimensions)	Click on the dimension line with the **Selection** tool and drag it to the desired location. • If the dimension standard does not have a fixed witness line length, the witness line endpoints remain stationary, and the witness line length is adjusted accordingly. • If the dimension standard has a fixed witness line length, the witness lines are not adjusted by default; to override the fixed length, press the Shift key while dragging the dimension line. To enter a specific value for the witness line length, use the Object Info palette. To enter a specific value for the offset, use the **Dim Off** field on the Object Info palette. 
Change the offset of the witness lines from the dimensioned object (linear, chain, and arc dimensions)	With the **Selection** tool, click on the gray dashed line at the endpoints of the witness lines and drag it to the desired location. The dimension line remains stationary, and the witness line length is adjusted accordingly. • If the dimension standard does not have a fixed witness line length, the Object Info palette shows that the witness lines have an offset override. • If the dimension standard has a fixed witness line length, the Object Info palette shows that the witness lines have a length override. To enter specific values for the offset or length of the witness lines, use the **Override** setting on the Object Info palette.  If the dimension has multiple override values for offset or length, each witness line has its own dashed line for editing. 

Action	Description
Modify the offsets of several dimension lines or witness lines at once (linear and chain dimensions)	With the **Selection** tool, select the dimensions to modify. Drag one of the dimension lines to the desired location; all selected dimension lines move the same distance. Drag one of the dashed gray lines that indicate the witness line offset to the desired location; the endpoints of all selected witness lines move the same distance. 

Modifying Dimensions

Measuring Distance

Distance can be measured without actually being recorded. This can be useful for placing objects or for reference.

Measuring in Units
Measuring in Degrees

Measuring in Units

The **Tape Measure** tool measures the distance between two or more points in the drawing area or in a sheet layer viewport, and temporarily displays the length in the Data bar. The tool shows the measurement between one click and the next, and it also keeps track of the cumulative length from the very first click.

Measurements display in the Data bar until the mouse is double-clicked. Note the measurements before the display clears.

 To measure a distance:

1. Click the **Tape Measure** tool from the appropriate tool set/palette.
   - All workspaces: Dims/Notes tool set
   - Spotlight workspace: Basic tool palette
2. Click where the first measurement is to start.
3. Move the cursor along the distance to measure.

   The Data bar displays the following information:

Data Bar Value	Description
L	Length (distance) from the previous point

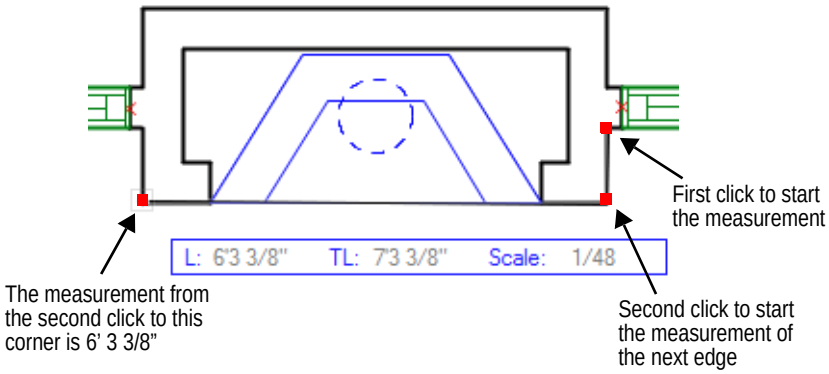
Data Bar Value	Description
TL	Total length (cumulative measurement) from the starting point
Scale	The scale set for the layer or sheet layer viewport where you clicked

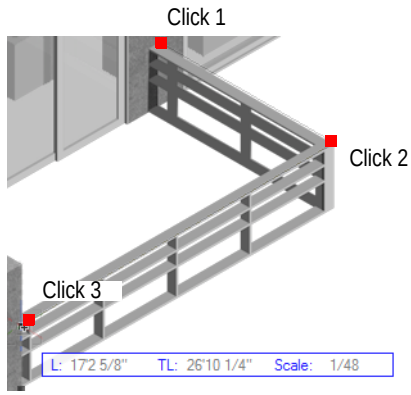
4. To continue measuring in a different direction or area, click to set the next starting point.

   The L measurement changes to zero.

5. Move the cursor to measure the next distance.

   The Data bar reflects the length from the previous point and the total cumulative length.





6. When the measurements are complete, note the total length.
7. Double-click to end the measurements.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Measuring Distance

Measuring in Degrees

The **Protractor** tool measures angles in the drawing, and temporarily displays the degrees measurement in the Data bar. There are two modes for the tool.

Mode	Description
Angle from Two Segments	Calculates the angle between two objects or object sides that are linear—rectangles (including rounded and rotated), lines, polylines, and polygons. This mode cannot, however, measure circles, ovals, or arcs. In addition, it cannot measure between parallel lines.
Angle from Three Points	Measures an angle between three points in the drawing area

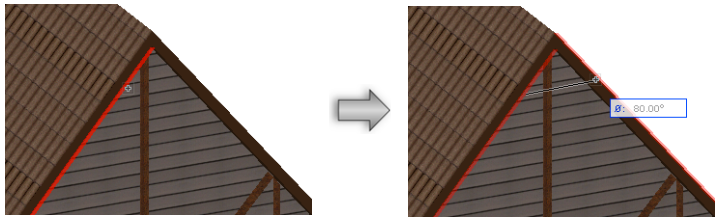
Angles Between Objects or Object Sides

To measure the angles between objects or object sides:

1. Click the **Protractor** tool from the appropriate tool set/palette, and select **Angle from Two Segments** mode.
   - All workspaces: Dims/notes tool set
   - Spotlight workspace: Basic tool palette
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Move the selection arrow over the first side of the angle to measure.
3. Click to select the side, and then move the cursor to the other angle side.

   The angle displays in the Data bar.

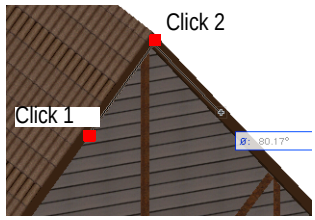


4. Note the angle, and then click to end the measurement.

Angles Between Three Points

To measure angles between three points:

1. Click the **Protractor** tool from the appropriate tool set/palette, and select **Angle from Three Points** mode.
   - All workspaces: Dims/notes tool set
   - Spotlight workspace: Basic tool palette
2. Click to set the first point.
3. Click to set the second point.
4. Move the cursor to the third angle point.



   The angle displays in the Data bar.

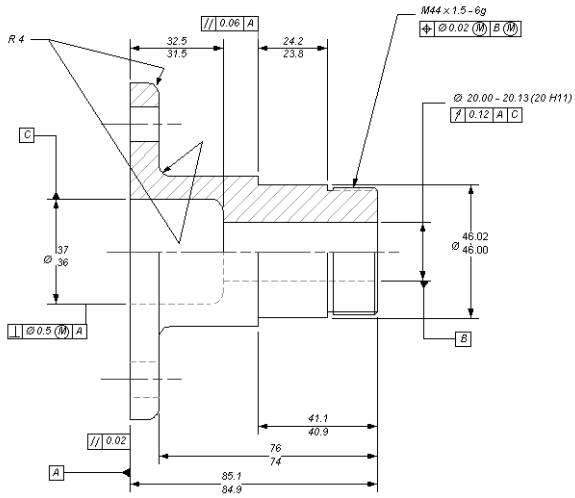
5. Note the angle, and then click to end the measurement.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Measuring Distance

Geometric Dimensioning and Tolerancing

The Vectorworks Design Series products provide the tools required for dimensioning a drawing according to ISO and ANSI/ASME geometric dimensioning and tolerancing standards.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



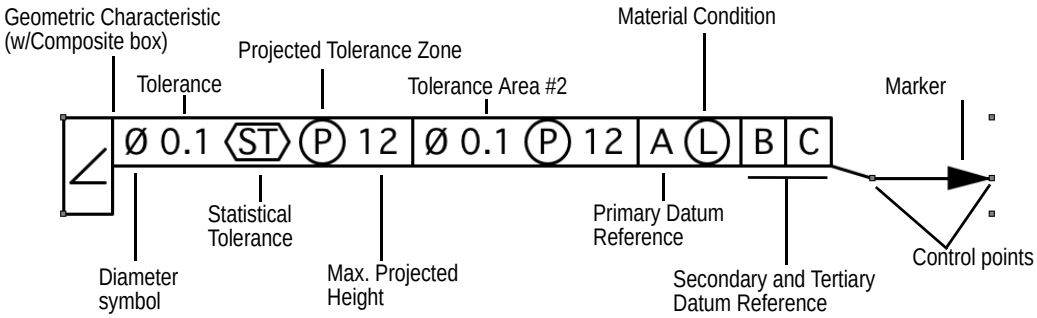
The Feature Control Frame, Datum Feature Symbol, and Datum Target Symbol are inserted into the Dimension class by default; control the appearance of these markers globally by editing the parameters of this class. To control the text characteristics (font size and style) globally, assign a text style to the class. You can also apply text styles or **Text** menu options to format individual markers.

For more information about editing class attributes and formatting text, see “Setting Class Properties” on page 179, “Formatting Text” on page 389, and “Using Text Styles” on page 393.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Feature Control Frame

A feature control frame consists of a geometric characteristic symbol, up to two tolerance areas, and up to three datum reference areas.



To insert a feature control frame:

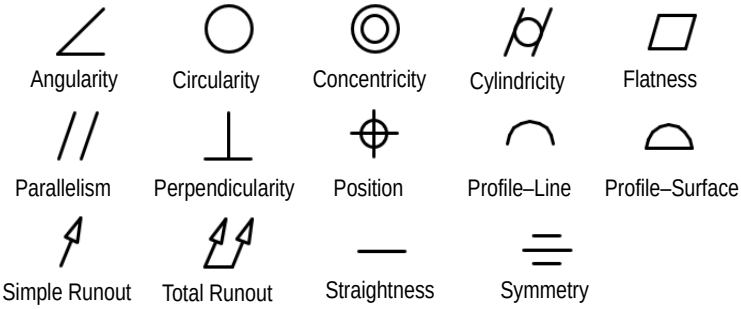
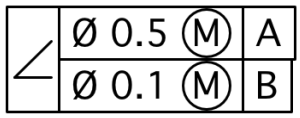
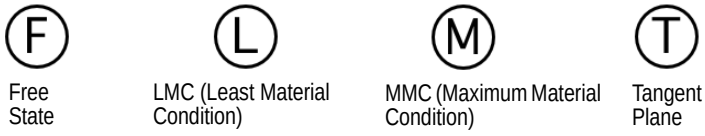
1. Click the **Feature Control Frame** tool from the Dims/Notes tool set.
2. Click on the drawing to insert the feature control frame. Click again to set the rotation of the object.

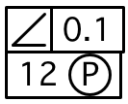
   If this is the first time the object has been inserted in this session, the Object Properties dialog box opens. If the Attributes palette has not been set to use end markers, select the marker style, size, and angle from the Object Properties dialog box.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Set the feature control frame parameters in the Object Info palette.

   If the leader is not immediately visible, drag it from the frame using the leader control points.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Geometric Characteristic	Select the geometric characteristic symbol, if any 
Composite Box	Select to increase the size of the geometric characteristic box; this is used when placing a second feature control frame 
Tolerance Area #1/#2	Select to include Tolerance Area #1 and/or Tolerance Area #2
Tolerance	Enter the tolerance value
Show Diameter Symbol	Select to display a diameter symbol along with the tolerance value Ø
Material Condition	Choose the material condition, if any 
Statistical Tolerance	Select to display a statistical tolerance symbol 
Projected Tolerance Zone	Select to display a projected tolerance zone symbol
Zone Height	Enter the maximum projected tolerance zone height when a projected tolerance zone symbol is displayed

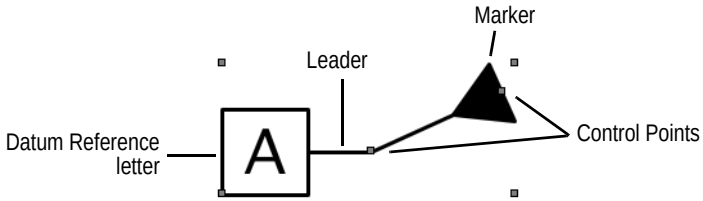
Parameter	Description
Place the PTZ symbol outside the FCF	Select to place the projected tolerance zone symbol outside and to the bottom of the feature control frame 
Primary/Secondary/ Tertiary Datum Reference	Select to include the primary, secondary, and/or tertiary datum reference
Reference	Enter the desired reference letter
Material Condition	Select a material condition, if any  LMC (Least Material Condition) MMC (Maximum Material Condition)
Leader Position	Select left or right

Once the feature control frame has been placed, its marker can be selected from the Attributes palette (see “Marker Attributes” on page 1102).

Geometric Dimensioning and Tolerancing
Marker Attributes

Datum Feature Symbol

A datum feature symbol consists of a marker, leader, and datum reference letter.



To insert a datum feature symbol:

1. Click the **Datum Feature Symbol** tool from the Dims/Notes tool set.
2. Click on the drawing to insert the datum feature symbol. Click again to set the rotation of the object.

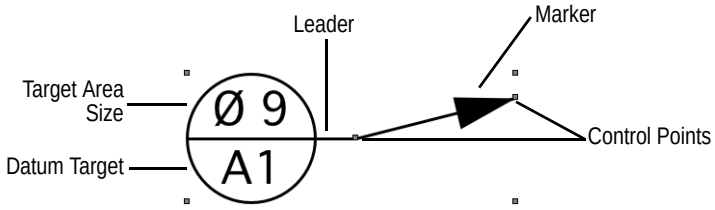
   If this is the first time the object has been inserted in this session, the Object Properties dialog box opens. If the Attributes palette has not been set to use end markers, select the marker style, size, and angle from the Object Properties dialog box.
3. Enter the datum reference letter (which corresponds to the Feature Control Frame reference letter).

Once the datum feature symbol has been placed, its marker can be selected from the Attributes palette (see “Marker Attributes” on page 1102). To control the text characteristics (font size and style) globally, assign a text style to the class. You can also apply text styles or **Text** menu options to format individual markers. See “Using Text Styles” on page 393 and “Formatting Text” on page 389.

Geometric Dimensioning and Tolerancing
Marker Attributes

Datum Target Symbol

A datum target symbol consists of a marker, leader, and target with datum target letter and target area size.

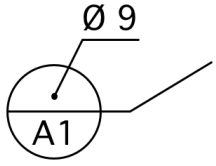


To insert a datum target symbol:

1. Click the **Datum Target Symbol** tool from the Dims/Notes tool set.
2. Click on the drawing to insert the datum target symbol. Click again to set the rotation of the object.

   If this is the first time the object has been inserted in this session, the Object Properties dialog box opens. If the Attributes palette has not been set to use end markers, select the marker style, size, and angle from the Object Properties dialog box.
3. Set the leader and marker position by dragging the symbol control points with the mouse. Specify the remaining datum target parameters in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Datum Target	Specify the datum target letter/number combination
Target Area Size	Enter the size of the target area
Show Diameter Symbol	Select to display the diameter symbol with the target area size Ø
Place target size outside of symbol	When necessary, the target size can be displayed outside the symbol; specify right or left **Placement** 

Once the datum target symbol has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).

Geometric Dimensioning and Tolerancing
Marker Attributes

Geometric Dimensioning and Tolerancing Note

The **Geom Dim and Tol Note** tool is used to insert a notation symbol and text specifying the dimension of round, square, counterbore, and/or countersink holes, and unit ratio for conical and/or flat tapers. Alternatively, a depth symbol and associated text can be inserted.

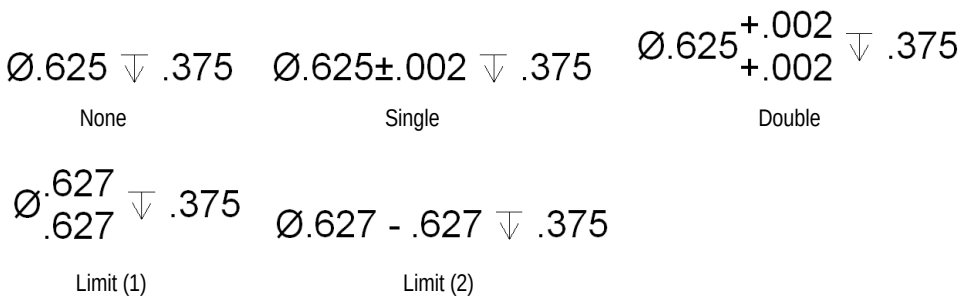
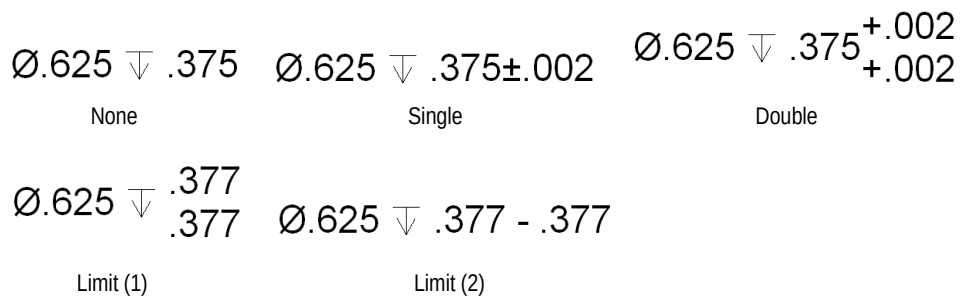
To insert a geometric dimensioning and tolerancing note:

1. Click the **Geom Dim and Tol Note** tool from the Dims/Notes tool set.
2. Click in the drawing to insert the note. Click again to set the rotation of the object.

   If this is the first time the object has been inserted in this session, the Object Properties dialog box opens. If the Attributes palette has not been set to use end markers, select the marker style, size, and angle from the Object Properties dialog box.
3. Specify the geometric dimensioning and tolerancing note parameters in the Object Info palette.

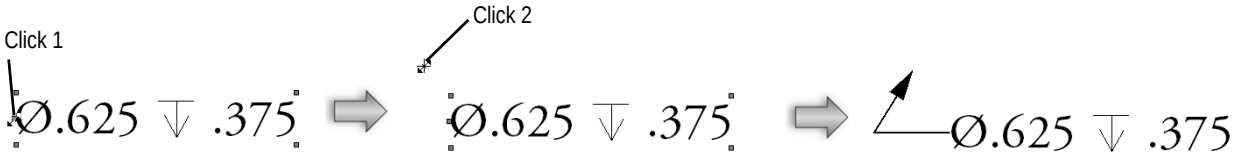
Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Precision	Select the decimal place rounding value for the hole size and depth
Note Type	Select the style of the note marker Ø.625 ↧ .375 Round Hole (and Depth) □ .625 ↧ .375 Square Hole (and Depth) ⌴ Ø.625 ↧ .375 Counterbore (and Depth) Ø.625 X 90° Countersink .3:1 Conical Taper .3:1 Flat Taper ↧ .375 Depth
Hole Size/ Taper	Enter the diameter for round, square, counterbore, and countersink holes; enter the unit ratio for conical and flat tapers

Parameter	Description
Tolerance	Sets whether the hole size/taper has a single, double, limited, or no tolerance 
Top/Bottom	Enter the value that is added to or subtracted from the base dimension to determine the upper/lower limit of the dimension
Depth/Angle	Enter the depth for round, square, and counterbore holes (if the depth is zero or less, it is not displayed); enter the angle for countersink holes
Tolerance	Sets whether the depth/angle has a single, double, limited, or no tolerance 
Top/Bottom	Enter the value that is added to or subtracted from the base dimension to determine the upper/lower limit of the dimension
Leader	Enter text to display before the dimension
Trailer	Enter text to display after the dimension
Leader Line Position	Select to place the dimension leader line to the right or left side of the object
Shoulder Length	Specifies the length of the leader line's shoulder

Once the geometric dimensioning and tolerancing note has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).

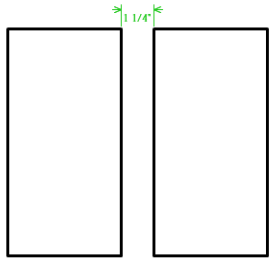
4. If desired, create and set a leader line using the **Selection** tool. Click the symbol control point. Crosshairs display to indicate the leader endpoint position. Click at the desired endpoint location; the leader line is drawn with the marker style, size, angle, and the shoulder length specified in the object preferences.



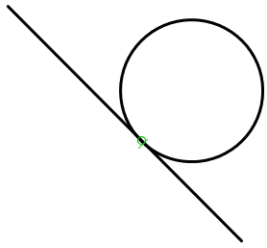
Marker Attributes

Parametric Constraints

Parametric constraints ensure that a drawing maintains its original precision. Parametric constraints maintain relationships between an object and world space, between two objects, or within the object itself. There are two types of parametric constraints: dimensional and geometric. Dimensional constraints maintain a measurable relationship by limiting the object's geometry to a particular value. Geometric constraints maintain a physical relationship by limiting the allowed orientation of objects.



Dimensional constraint maintaining the horizontal distance between two objects



Geometric constraint maintaining tangency between a line and a circle

Parametric constraints can be placed on all 2D objects. They cannot be placed on 3D objects except for walls, symbols, and plug-in objects that have 2D components. Multiple constraints can be applied to an object. Parametric constraints can be placed across layers as long as both layers are of the same scale, and Layer Options are set to **Show/Snap/ Modify Others**.

Constraints must always be placed in the plane defined by the object or objects being constrained; when multiple objects are constrained, they must be co-planar to create the constraint. The constraint on an object or objects changes planes with the object, if the object changes its plane.

Constraints attached to a single object move along with the object even if the object is copied or cut and pasted. When only one of a pair of constrained objects is duplicated or copied or cut and pasted, the constraint is removed.

When a parametric constraint is placed, green constraint indicators are drawn for the object(s) involved. To hide indicators, deselect **Show parametric constraints** in the Display tab of the Vectorworks preferences. Alternatively, select **View > Show > Show or Hide Constraints**; the command toggles between displaying or hiding constraints, as appropriate.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Dimensional Constraints

Dimensional constraints maintain a measurable relationship. They resemble standard dimensions when placed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Constrain Angle

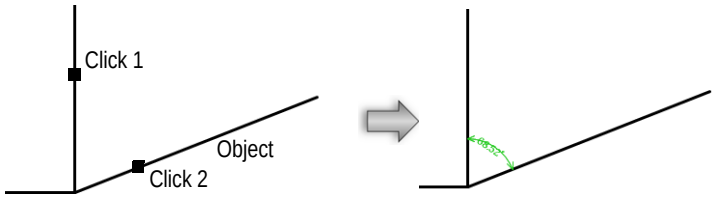
Constrain the angular relationship between separate objects or line segments of a single object. If one object or segment is rotated, the object or segment it is constrained to adjusts to maintain the angle.

To constrain the angle between objects or line segments of a single object:

1. Click the **Constrain Angle** tool from the Dims/Notes tool set.
2. Click on one of the two objects or line segments to be constrained.

   The cursor switches to the bull's-eye cursor.
3. Click on the second object or line segment to be constrained.

   A green angle constraint is drawn between the two objects or line segments.



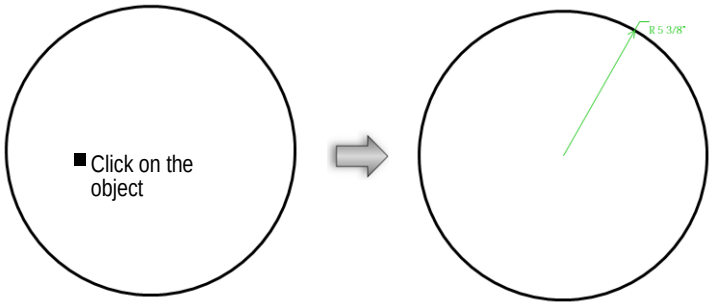
Constrain Radius

Constrain the radius of a single arc or circle. If the arc or circle is accidentally resized, the constraint prevents the operation, preserving the radius.

To constrain the radius of an arc or circle:

1. Click the **Constrain Radius** tool from the Dims/Notes tool set.
2. Click on the arc or circle to be constrained.

   A green radius constraint is drawn on the object.



The **Constrain Radius** tool does not work on quarter arcs.

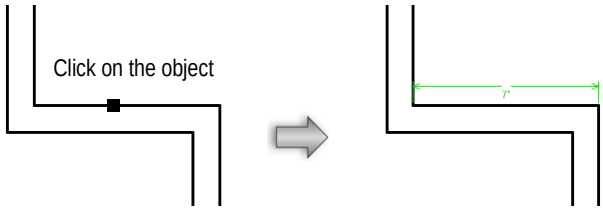
Constrain Horizontal Distance

Constrain the horizontal distance of an edge of an object, a line segment, or between two points. If an object resize is attempted, the constraint prevents the operation, preserving the original horizontal distance. When the constraint is on two different objects, if one object is modified, the object it is constrained to moves to remain at the same constrained horizontal distance.

To constrain the horizontal distance of an edge of an object or a line segment:

1. Click the **Constrain Horiz Distance** tool from the Dims/Notes tool set.
2. Click on the object to be constrained.

A green horizontal distance constraint is drawn on the object.

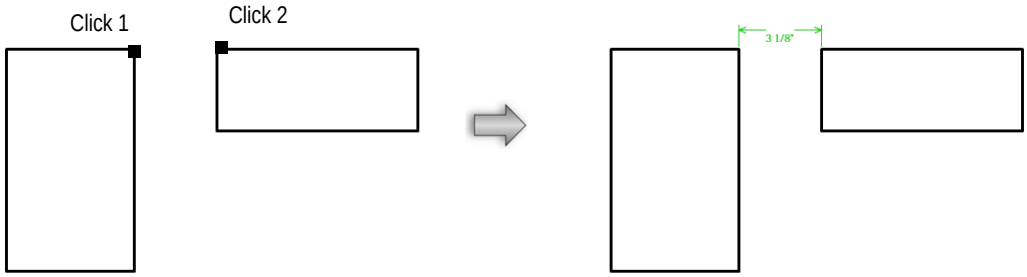


To constrain the horizontal distance between two points:

1. Click the **Constrain Horiz Distance** tool from the Dims/Notes tool set.
2. Click on the first point to be constrained.

   The cursor switches to the bull's-eye cursor.
3. Click on the second point to be constrained.

   A green horizontal distance constraint is drawn between the two points.



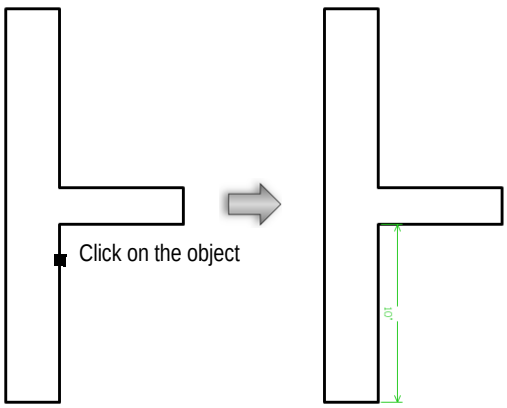
Constrain Vertical Distance

Constrain the vertical distance of an edge of an object, line segment, or between two points. If an object resize is attempted, the constraint prevents the operation, preserving the original vertical distance. When the constraint is on two different objects, if one object is modified, the object to which it is constrained moves to remain at the same constrained vertical distance.

To constrain the vertical distance of an edge of an object or line segment:

1. Click the **Constrain Vertical Distance** tool from the Dims/Notes tool set.
2. Click on the object to be constrained.

   A green vertical distance constraint is drawn on the object.

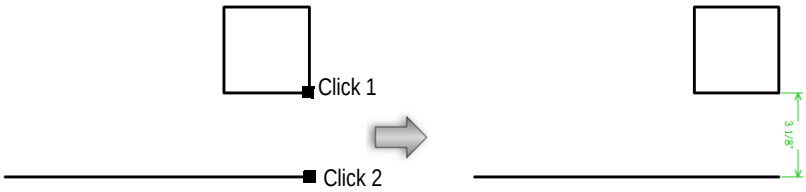


To constrain the vertical distance between two points:

1. Click the **Constrain Vertical Distance** tool from the Dims/Notes tool set.
2. Click on the first point to be constrained.

   The cursor switches to the bull's-eye cursor.
3. Click on the second point to be constrained.

   A green vertical distance constraint is drawn between the two points.



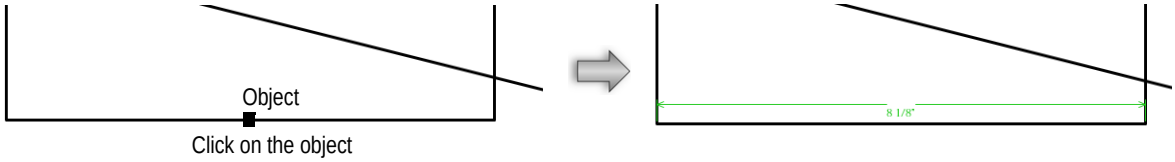
Constrain Distance

Constrain the distance of an edge of an object, line segment, or between two points regardless of the angle. If an object is accidentally resized, the constraint prevents the operation, preserving the original distance. When the constraint is on two different objects, if one object is modified, the object to which it is constrained moves to remain at the same constrained distance.

To constrain the distance of an edge of an object or line segment:

1. Click the **Constrain Distance** tool from the Dims/Notes tool set.
2. Click on the object to be constrained.

   A green horizontal distance constraint is drawn on the object.

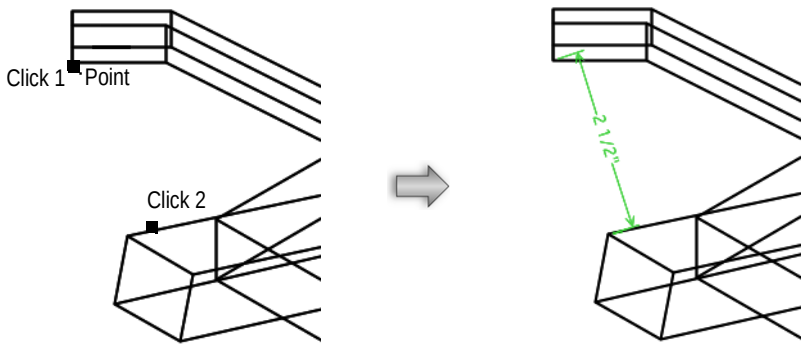


To constrain the distance between two points:

1. Click the **Constrain Distance** tool from the Dims/Notes tool set.
2. Click on the first point to be constrained.

   The cursor switches to the bull's-eye cursor.
3. Click on the second point to be constrained.

   A green distance constraint is drawn between the two points.



Geometric Constraints

Geometric constraints preserve the geometric properties of objects.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Constrain Horizontal-Vertical
Constrain Parallel
Constrain Perpendicular
Constrain Collinear
Constrain Coincident
Constrain Concentric
Constrain Tangent

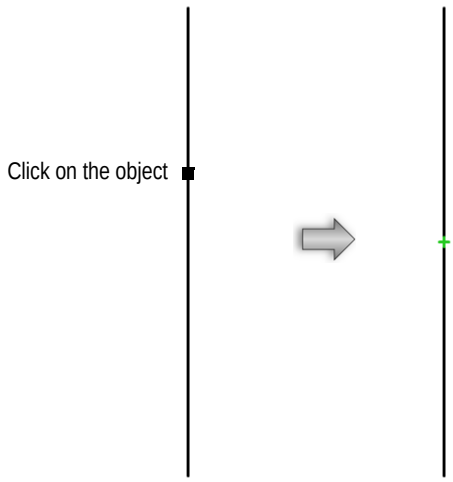
Constrain Horizontal-Vertical

Constrain a linear object to remain horizontal or vertical. Once constrained, the object cannot be rotated to any other position. The object can be resized in length, but it always remains horizontal or vertical.

To constrain an object to remain horizontal-vertical:

1. Click the **Constrain Horiz-Vertical** tool from the Dims/Notes tool set.
2. Click on the linear object to be constrained.

   A green horizontal-vertical constraint is drawn on the object.



If the object is diagonal when the constraint is placed, it rotates to become vertical or horizontal, depending on which angle it is closest to.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Constrain Parallel

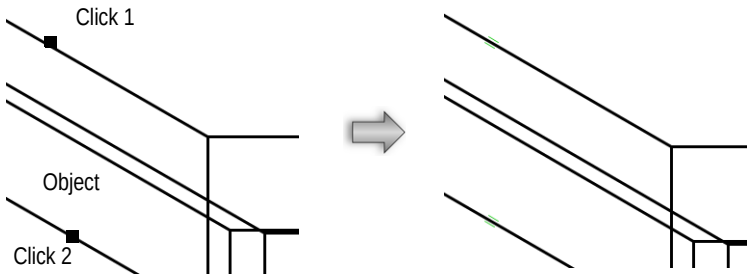
Constrain linear objects or line segments to be parallel to one another. If one object is rotated, the object constrained to it rotates to remain parallel to the first object. Lines do not need to be parallel when placing the constraints; the first line rotates to match the angle of the second line.

To constrain linear objects or line segments to be parallel:

1. Click the **Constrain Parallel** tool from the Dims/Notes tool set.
2. Click on the line to constrain.

   The cursor switches to the bull's-eye cursor.
3. Click on the line to be constrained.

   Green parallel constraint indicators are drawn around the two lines.



Constrain Perpendicular

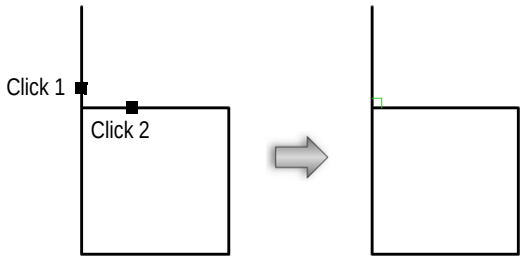
Constrain linear objects or line segments to be perpendicular to one another. If one line is rotated, the line it is constrained to adjusts to remain perpendicular to the first line. Lines do not need to be perpendicular when placing the constraints; the first line rotates to become perpendicular to the second line.

To constrain lines to be perpendicular:

1. Click the **Constrain Perpendicular** tool from the Dims/Notes tool set.
2. Click on the line to constrain.

   The cursor switches to the bull's-eye cursor.
3. Click on the line to be constrained.

   A green perpendicular constraint is drawn, connecting the two lines.



Constrain Collinear

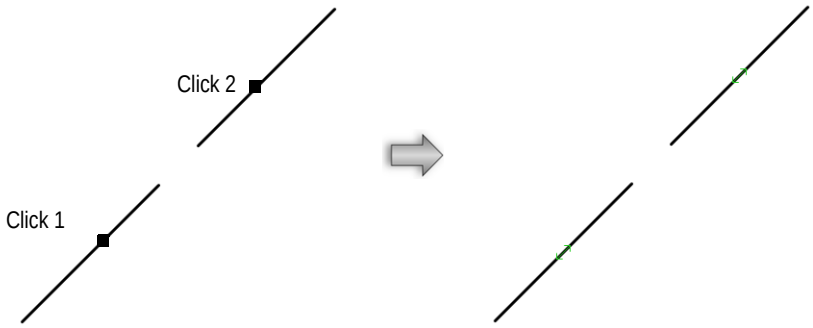
Constrain the collinearity between two linear objects. If one line is moved, the line it is constrained to adjusts to remain aligned. Lines do not need to be aligned when placing the constraints; the first line moves to become collinear to the second line.

 To constrain the collinearity between two lines:

1. Click the **Constrain Collinear** tool from the Dims/Notes tool set.
2. Click on the line to constrain.

   The cursor switches to the bull's-eye cursor.
3. Click on the line to be constrained.

   Green collinear constraint indicators are drawn on the two lines.



Constrain Coincident

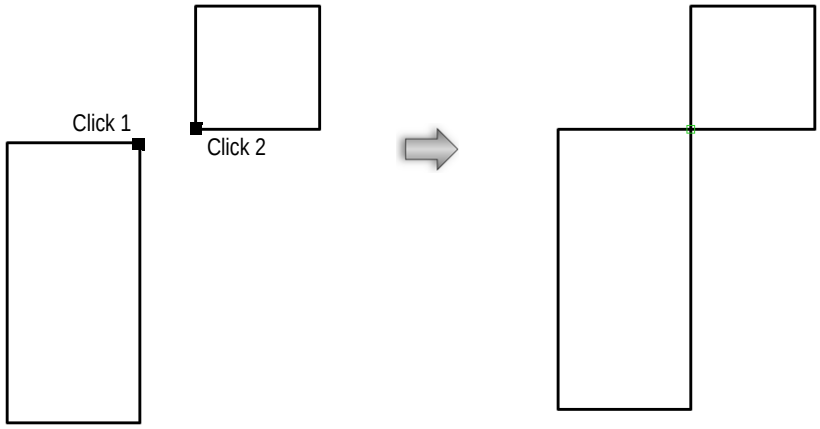
Constrain two selected points to remain attached. If one object is moved, the object it is constrained to adjusts to maintain the connection. The first point stretches to connect to the second point, if necessary.

 To constrain the connection between two points:

1. Click the **Constrain Coincident** tool from the Dims/Notes tool set.
2. Click on the point to constrain.

   The cursor switches to the bull's-eye cursor.
3. Click on the point to be constrained.

   A green coincident constraint is drawn where the two points touch.



Constrain Concentric

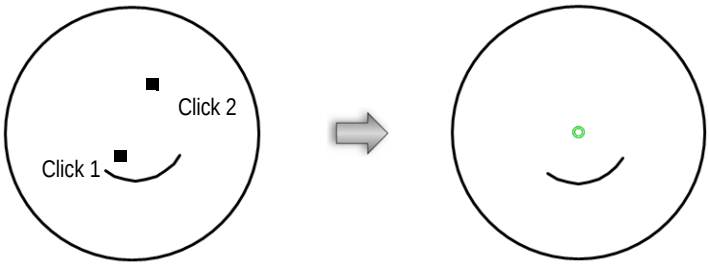
Constrain circles and arcs concentrically. If a circle or arc is moved, the circle or arc it is constrained to moves so that their centers remain aligned. Circles and arcs do not need to be concentric when placing the constraints; the first object moves so that its center aligns to the second object's center.

To constrain two circles or arcs concentrically:

1. Click the **Constrain Concentric** tool from the Dims/Notes tool set.
2. Click on the object to constrain.

   The cursor switches to the bull's-eye cursor.
3. Click on the object to be constrained.

   A green concentric constraint is drawn at the center of the two objects.



Constrain Tangent

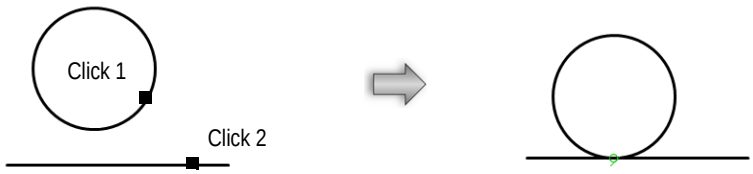
Constrain a circle, arc or line to be tangent to another circle or arc. If one object is moved, the other object it is constrained to adjusts to maintain the tangency. Objects do not need to be tangent to one another when placing the constraints; the first object moves to become tangent to the second.

To constrain a circle, arc or line to be tangent to a circle or arc:

1. Click the **Constrain Tangent** tool from the Dims/Notes tool set.
2. Click on the circle or arc to constrain.

   The cursor switches to the bull's-eye cursor.
3. Click on the circle, arc or line to be constrained.

A green tangent constraint is drawn at the tangent point of the two objects.



Editing Parametric Constraints

When a constrained object is deleted, the parametric constraint attached to it is also removed. To remove the constraint without removing the attached object, use the **Edit Constraints** command.

A situation may arise where the value of a dimensional constraint needs to be changed. A wall with a horizontal distance constraint value of 2’ 6” may at a later point in the project need to be changed to 5’ 6”. This type of edit is also accomplished using the **Edit Constraints** command.

Individual constraints can be selected from a list of all constraints currently applied to a selected object. The selected constraint changes color to indicate which one is about to be edited. Geometric constraints, where appropriate, along with changing the constraint color, also show the connection between the two constrained objects for clarity.

Constraint errors related to editing operations that conflict with current constraints are solved by either removing the constraint, or canceling the editing operation.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Deleting Parametric Constraints
Changing a Dimensional Constraint Value
Parametric Constraint Errors

Deleting Parametric Constraints

You can select which constraints to delete or delete all constraints at once. Constraints can be removed or edited for multiple selected objects.

To select which parametric constraint to delete:

1. Select the object with the constraint.
2. Select **Modify > Edit Constraints**.

   The Edit Constraints dialog box opens.
3. Select the constraint to be removed from the Constraints list.

   The selected constraint changes color.
4. Click **Delete** to remove the constraint.
5. Click **OK**.

To delete all parametric constraints:

1. Select the object(s) with the constraints.
2. From the context menu, select **Remove Constraints**. Alternatively, select **Delete All** from the Edit Constraints dialog box.

   Using the **Remove Constraints** context menu command will also disassociate any associated dimensions.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Parametric Constraints
Changing a Dimensional Constraint Value

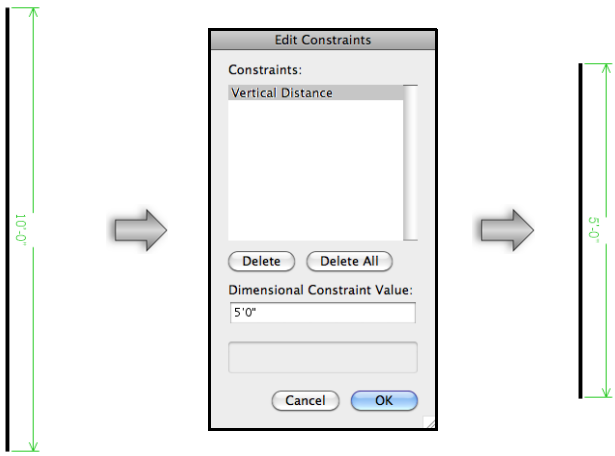
Parametric Constraint Errors

Changing a Dimensional Constraint Value

To change a dimensional constraint value:

1. Select the object with the constraint.
2. Select **Modify > Edit Constraints**.

   The Edit Constraints dialog box opens.



3. In the **Dimensional Constraint Value** field, enter the new dimension, and click **OK**.

   The dimensional value is updated and the object or objects are adjusted.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Parametric Constraints
Deleting Parametric Constraints
Parametric Constraint Errors

Parametric Constraint Errors

Occasionally, an editing operation cannot be performed due to conflicting or unsolvable constraints placed on the objects involved. When this occurs, an alert dialog box opens which allows the unsolvable constraints to be previewed and, if needed, removed.

To solve conflicting constraint errors:

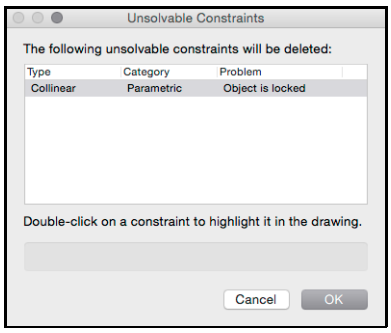
1. When an unsolvable constraint occurs, an alert dialog box opens. Decide how to resolve the conflicting situation.

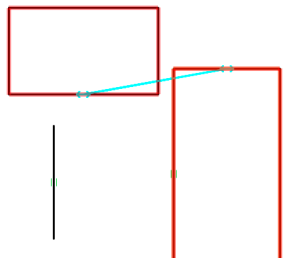
Click to show/hide the parameters.

Parameter	Description
No	Continues with the editing operation, removing the conflicting constraints
Yes	Cancels the editing operation, preserving constraints
Preview	Opens the Unsolvable Constraints dialog box, listing the problem constraints and allowing a preview of each constraint

2. Select **Preview constraints that will be selected**, and then click **Yes** to preview the unsolvable constraints.

   The Unsolvable Constraints dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~





3. Double-click on an unsolvable constraint to preview it in the drawing. The unsolvable constraint displays with a different color.
4. Click **OK** to remove the constraints and perform the editing operation. Click **Cancel** to retain the constraints and cancel the editing operation.

Editing Parametric Constraints
Deleting Parametric Constraints
Changing a Dimensional Constraint Value

Annotation

Inserting Leader Lines

The Vectorworks Fundamentals product provides a basic leader line object. In a Vectorworks Design Series workspace, the leader line includes additional parameters.

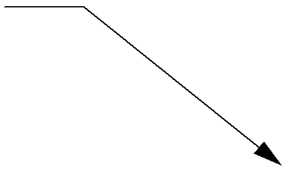
Inserting a Simple Leader Line

The **Leader Line Simple** tool inserts a basic leader line. Two modes are available.

Mode	Description
Constrained	Constrains the line to be vertical, horizontal, and 30° or 45° from vertical or horizontal in any direction
Unconstrained	Draws the line at any angle Press and hold the Shift key to snap the line to predetermined angles

 To insert a leader line symbol:

1. Click the **Leader Line Simple** object from the Basic tool set and select the **Constrained** or **Unconstrained** mode from the Tool bar.
2. Click and move the mouse to define the leader line shoulder length. Click and move the mouse to define the leader line, and click to set the end point.



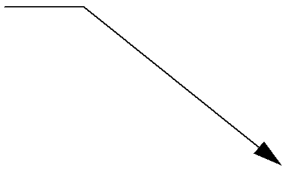
Once the leader line has been placed, its marker can be selected from the Attributes palette (see “Marker Attributes” on page 1102).

D Inserting a Design Series Leader Line

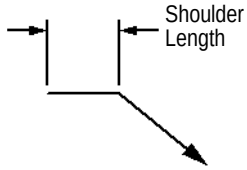
 To insert a leader line symbol:

Click the **Leader Line** object from the Basic tool set.

If this is the first time the object has been inserted in this session, the Object Properties dialog box opens. Set the default properties for the object and click **OK**. The properties can be edited after placement from the Object Info palette.



Click to show/hide the parameters.

Parameter	Description
Shoulder Length	Sets the length of the leader line's shoulder 
Arrows	Sets whether to have an arrow on the leader line and if so, the side from which it extends
Marker Style	Select a marker style from the list
Marker Size/Marker Angle	Sets the size of the marker and the angle of the leader line

If there are several leader line objects, use the **Align/Distribute Leader Lines** command to improve readability.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Marker Attributes
Aligning and Distributing Leader Lines
Using Notation Objects

Using Notation Objects

Several tools in the Vectorworks Fundamentals product add set information to a drawing with preformatted objects. Vectorworks Design Series products offer tools for additional notation objects.

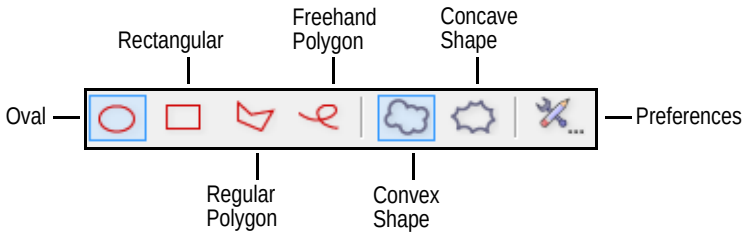
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

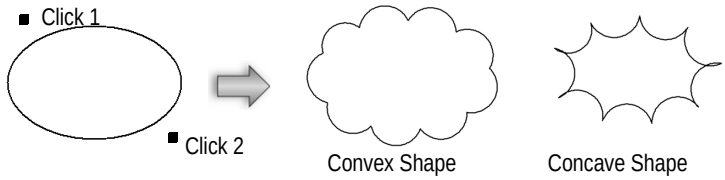
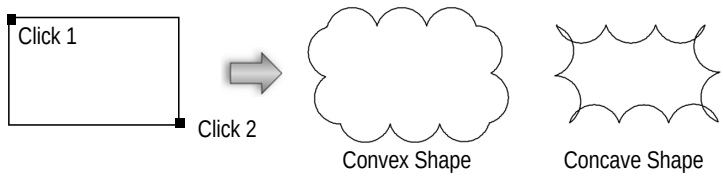
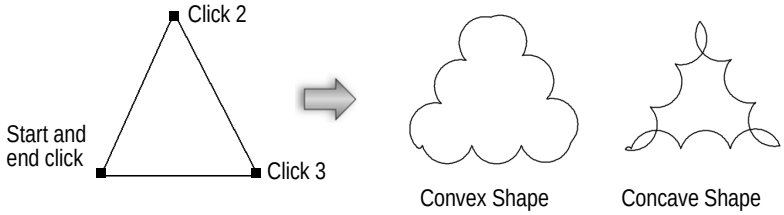
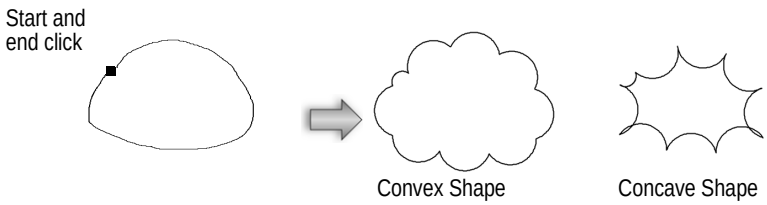
Creating Revision Clouds
Creating Drawing Labels
Creating Reference Markers
Creating Data Stamps
North Arrow
Revision Marker
Elevation Benchmark
Indicating the Slope of Surfaces
Inserting a Scale Bar
Creating Grid Bubbles
Section Lines and Section-Elevation Markers
Inserting a Symmetry Label

Creating Revision Clouds

Use the **Revision Cloud** tool to identify a section of a drawing that has changed. Insert a revision cloud in an area of the drawing, or around an entire portion of the drawing, if appropriate.

For the Vectorworks Design Series products, revision clouds can also be created by first drawing a polyline and then selecting the **Create Objects from Shapes** command (see "Creating Objects from Shapes" on page 277).



Mode	Description
Oval	Inserts the revision cloud around the perimeter of a preview oval. Click to set the start point, move the cursor in the desired direction, and click to set the end point. Constrain the oval to 45 degrees to draw a circle. 
Rectangular	Inserts the revision cloud around the perimeter of the preview rectangle. Click to set the start point, move the cursor in the desired direction, and click to set the end point. Constrain the rectangle to 45 degrees to draw a square. 
Regular Polygon	Inserts the revision cloud around the perimeter of the specified vertices. Click to set the start point (first vertex), click at the desired location for each subsequent vertex, and either click at the starting vertex to close the polygon or double-click at the final vertex to create an open polygon. If the polygon is open, the cloud is completed based on the outline. 
Freehand Polygon	Inserts the revision cloud around the perimeter of the specified vertices. Click to set the start point and click-drag to draw the freehand polygon. If the polygon is open, the cloud is completed based on the outline. 

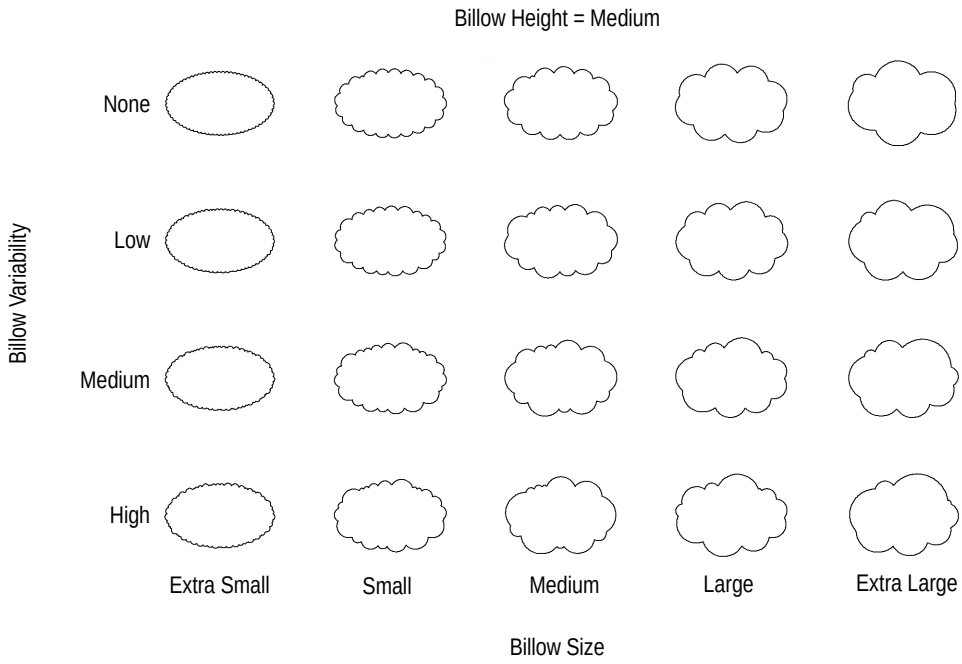
To create a revision cloud:

1. Click the **Revision Cloud** tool from the Dims/Notes tool set.
2. Click the **Preferences** button on the Tool bar to define the revision cloud properties. The properties can also be set after placement, from the Object Info palette. Click **OK**.
3. Select the revision cloud mode and select to draw the cloud with either a convex or concave shape. The convex shape draws billows outward from the preview image or from the specified vertices. The concave shape draws billows inward from the preview image or from the specified vertices.
4. Click to draw the revision cloud according to the specified mode.

Click to show/hide the parameters.

Parameter	Description
Billow Size	Select the relative billow size from extra small to extra large, or select Custom to specify a billow size
Billow Radius	For a custom billow size, sets the billow size
Billow Variability	Select the variability (the variation between the smallest and largest billows)
Billow Height	Select the relative billow height (small, medium, or large)
Billow Type	Select a convex or concave billow type
Vertex Parameters	Edits the revision cloud vertices; see “Editing Vertex-Based Objects” on page 1002

The following illustration demonstrates the effects of varying the billow size and billow variability parameters.



Creating Drawing Labels

The **Drawing Label** tool is a point object that provides descriptive information for the drawing.

By default, a label object includes a title, the scale of the drawing, and an automatically assigned drawing number. The number of the sheet that contains the drawing can also be included in the label.

Many of these values default, depending on where the label was created: on a design layer, or in a viewport annotation, for example.

 To create a drawing label:

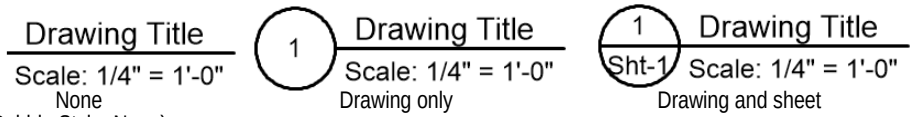
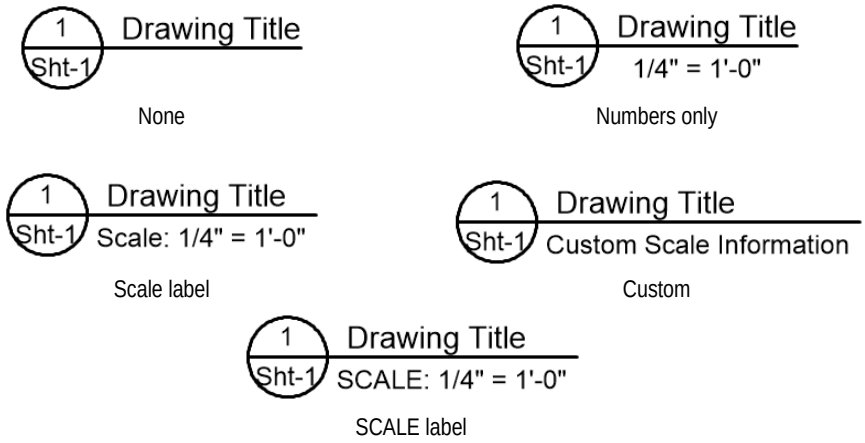
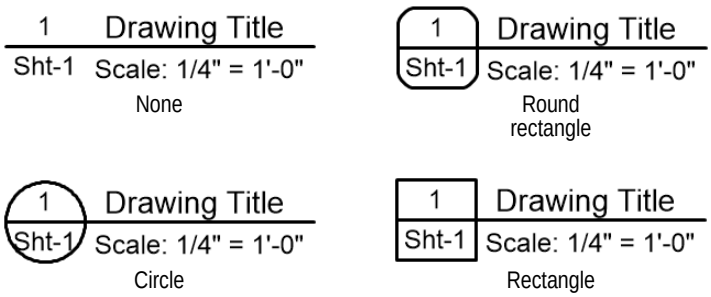
1. Ensure that the appropriate layer is active.
2. Click the **Drawing Label** tool from the Dims/Notes tool set.
3. Click on the drawing at the desired location to place a drawing label object.
4. Click again to set the rotation.

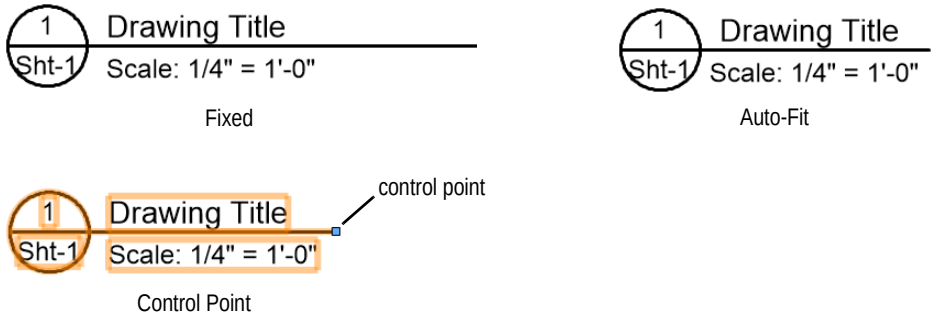
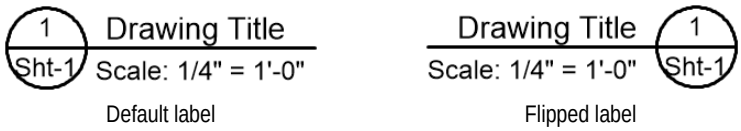
   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.

   Drawing label objects maintain a constant size regardless of the drawing scale.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Drawing Title	Specifies the title text that displays above the label's horizontal ruling line; defaults to the file name (on a design layer), to nothing (on a sheet layer), or to the **Drawing Title** (in a viewport), but it can be edited. If **Use Automatic Drawing Coordination** is enabled in document preferences (Vectorworks Design Series required), and the label is in a viewport, changing this field will update the viewport's drawing title, and vice versa.
Title Alignment	Specify the alignment of the **Drawing Title** in reference to the line. Select between left, center, and right alignment. For center and right alignment, if the drawing label has a bubble style and the line length mode is not Auto-Fit, text may overlap the bubble if text is longer than the line.
Drawing Number	Identifies this drawing on the current sheet. This value displays in the top half of the label bubble, unless **Number Style** is set to None. If the label is in a viewport or sheet layer, the program automatically numbers items and updates this field; this number must be unique on this layer. If the label is on a design layer, enter an identifier manually.
Sheet Number	Identifies the sheet that contains the drawing. This value displays in the bottom half of the label bubble, if **Number Style** is set to Drawing and Sheet. If the label is on a sheet layer, this field defaults to the Sheet Number. If **Use Automatic Drawing Coordination** is enabled in document preferences (Vectorworks Design Series required), and the label is on a sheet layer, this field cannot be edited and always contains the Sheet Number.

Parameter	Description
Number Style	Specifies which drawing identifier numbers (from the **Drawing Number** and **Sheet Number** fields) display in the label 
Scale Display Style	Specifies which scale information displays beneath the label's horizontal ruling line 
Custom Scale	If **Scale Display Style** is set to Custom, enter information to display in the scale area
Scale Alignment	Aligns the **scale display** in reference to the line. Select between left, center, or right alignment. For center and right alignment, if the drawing label has a bubble style and the line length mode is not Auto-Fit, text may overlap the bubble if text is longer than the line.
Use Architectural Scale	Select this option to use the US Architect scale style (as in 1/4" = 1'- 0"); deselect the option to use the Engineering style (as in 1:48)
Bubble Style	Select the style of the label bubble graphic, which can contain the **Drawing Number** and **Sheet Number** that identify the drawing 
Draw Full-width Bubble	Select this option to extend the bubble graphic to the full length of the ruling line; not available if **Bubble Style** is set to None
Title / Scale / Drawing# / Sheet# Text Size	Select the font sizes for the various elements of the drawing label
Bubble Scale Factor	Specifies the label's bubble size; enter a larger value to increase the size (the number must be greater than zero)

Parameter	Description
Line Length Mode	Sets the length of the drawing label's horizontal ruling line. A fixed length ruling line is set by the **Printed Length** value. An auto-fit ruling line is set to the length of the longest text bounding box, either above or below the ruling line. A control point ruling line is set by the location of its control point; click the control point to select it and click in the desired location in the drawing to adjust the ruling line length or to flip the ruling line to the opposite side. 
Printed Length	For a drawing label using a fixed length ruling line, sets the length of the drawing label on the printed page
Flip	Select this option to move the bubble (including the **Drawing Number** and **Sheet Number**) to the right side of the label: 
Rule Offset	Specifies the vertical offset between the label's bubble and horizontal ruling line on the printed page (0 is centered); enter a negative number to move the line, title, and scale below the center line of the bubble
Title Margin	Specifies the vertical distance between the label's horizontal ruling line and the text above it. Enter a positive value to move the text further above the ruling line, or a negative value to move the text closer to the ruling line.
Scale Margin	Specifies the vertical distance between the label line and the text below it. Enter a positive value to move the text further above the ruling line, or a negative value to move the text closer to the ruling line.
Show Drawing Title	Select to display the drawing title; if deselected, both the drawing title and the ruling line are not displayed
Show Scale with Drawing Title	Select to display the scale and drawing title on the same line; this option is only available when **Show Drawing Title** is selected
Position Scale on Center Line	Select to position the scale in place of the drawing title; this option is only available when **Show Drawing Title** is deselected

5. Set the line attributes as needed using the Attributes palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Notation Objects
The Attributes Palette
Formatting Text
Using Text Styles
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Reference Markers

The **Reference Marker** tool displays the drawing number and sheet number of the referenced drawing. Select from a variety of configurations.

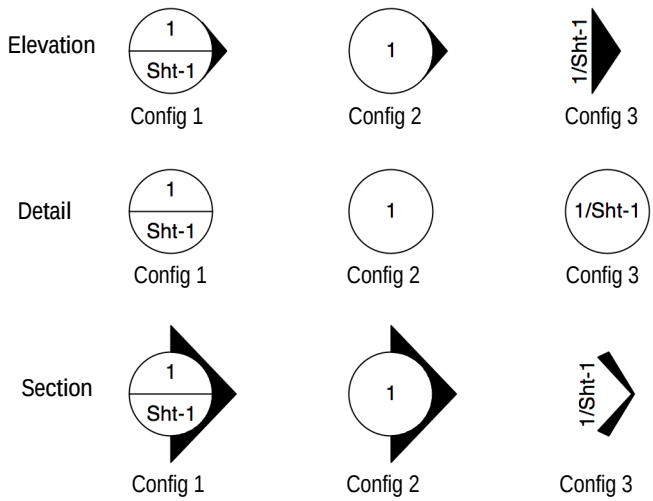
To create a reference marker:

1. Ensure that the appropriate layer is active.
2. Click the **Reference Marker** tool from the Dims/Notes tool set.
3. Click on the drawing at the desired location to place a reference marker object.
4. Click again to set the rotation.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.

   Reference marker objects maintain a constant size regardless of the drawing scale.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
2D Scale Factor	Specifies the marker size; increase the scale value to obtain a larger marker
Type/Config	Select one of the types and configurations from the list 
Drawing Number	Specifies the drawing number of the item the marker references
Sheet Number	Specifies the sheet number of the item the marker references
Separator	For Config 1 configurations, specifies the separator between the **Drawing Number** and **Sheet Number** information
Arrow Angle	For some configurations, specifies the angle of the marker arrow
Arrow Fill	Select a filled or unfilled arrow style from the list

5. Set the line attributes as needed using the Attributes palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Notation Objects
The Attributes Palette
Formatting Text
Using Text Styles

Creating Data Stamps

The **Data Stamp** tool displays the date, time and file name of the current drawing. Select from a variety of configurations.

 To create a data stamp:

1. Click the **Data Stamp** tool from the Dims/Notes tool set.
2. Click on the drawing at the desired location to place a data stamp.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**.
3. Click again to set the rotation.

   The data stamp parameters can be edited from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Show Date	Select to display the current date
Show Time	Select to display the current time in hours, minutes, and seconds
Show File Name	Select to display the drawing's file name
Draw Box	Select to display a border around the data stamp
Use Labels	Select to display date, time, and file name labels placed before the data
Date Display Style	Select the format for the date display. Options include: Short - 1/1/15 Medium - Jan 1, 2015 Long - January 1, 2015 Medium w/ Weekday - Mon, Jan 1, 2015 Long w/ Weekday - Monday, January 1, 2015
Text Size	Select a font size for the data; the font itself can be changed from the **Text** menu
Data Style	Select a font style for the data: plain, bold, italic, or bold italic
Label Style	Select a font style for the data labels: plain, bold, italic, or bold italic
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Margin Spacing	Specify the space between the text and the surrounding border
Label Size Factor	Specify the label size relative to the data size
Update	Click **Update** to manually update the data stamp date, time, or file name prior to printing it or exporting it, so that it reflects the most current data

The date and time formats are determined by each Vectorworks application user's local computer settings.

Using Notation Objects
The Attributes Palette
Formatting Text
Using Text Styles

North Arrow

The **North Arrow** tool indicates the drawing orientation; several different styles can be selected. The deviation from true magnetic north can be displayed on some of the configurations. If the design layer is georeferenced, the object is rotated to point in the proper direction.

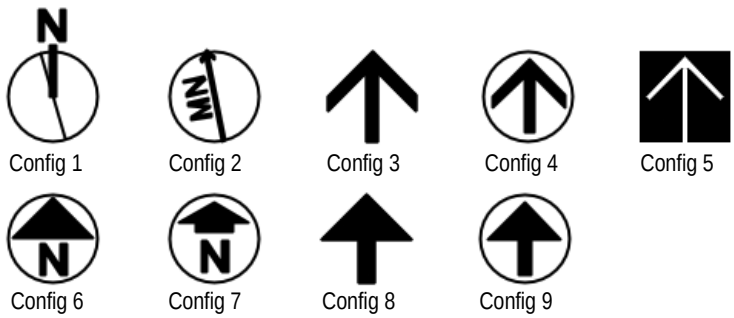
To place a north arrow object:

1. Click the **North Arrow** tool from the Dims/Notes tool set.
2. Click on the drawing at the desired location to place a north arrow object.
3. Click again to set the rotation.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.

   North arrow objects maintain a constant size regardless of the drawing scale.

Click to show/hide the parameters.

Parameter	Description
2D Scale Factor	Specifies the marker size; increase the scale value to obtain a larger marker
Config	Select the arrow style from the list 
Arrow Graphics	Select an arrow style from the list; arrow styles include filled and unfilled versions
Show MN Declination	Select to add the deviation from true magnetic north for some styles
Declin Deg Offset	Specifies the degree of offset from true north

Using Notation Objects

Revision Marker

The **Revision Marker** tool displays the drawing revision number.

 To place a revision marker object:

1. Click the **Revision Marker** tool from the Dims/Notes tool set.
2. Click on the drawing at the desired location to place a revision marker object.
3. Click again to set the rotation.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.

   Revision marker objects maintain a constant size regardless of the drawing scale.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393.
2D Scale Factor	Specifies the marker size; increase the scale value to obtain a larger marker
Revision Number	Specifies the revision number

Using Notation Objects

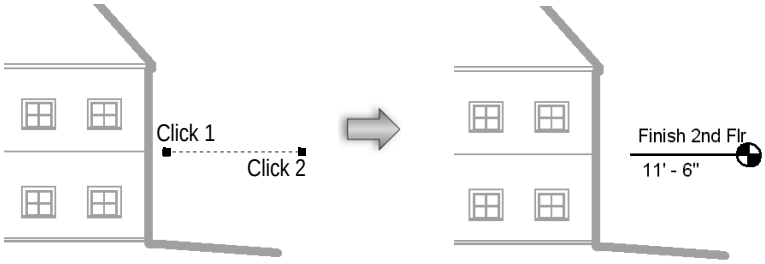
Elevation Benchmark

The **Elevation Benchmark** tool represents different levels in elevation drawings; two different elevation benchmark styles are available: ISO and US.

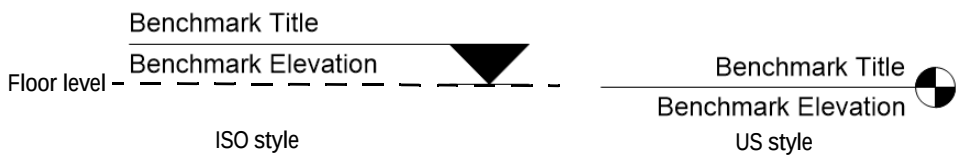
 To place an elevation benchmark object:

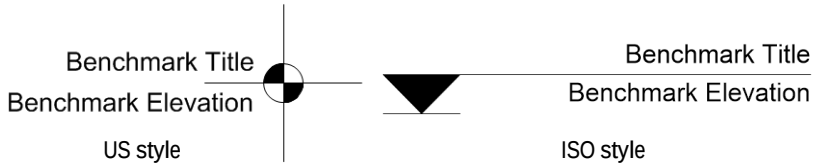
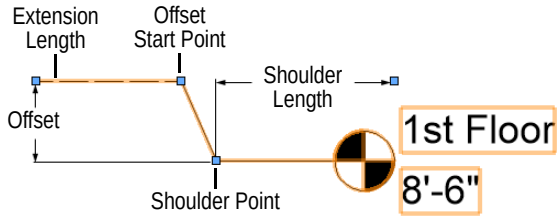
1. Click the **Elevation Benchmark** tool from the Dims/Notes tool set.
2. Click on the drawing at the desired location to place an elevation benchmark linear object.
3. Click to set the length and rotation of the elevation benchmark.

   If this is the first time the object is inserted in this session, the Object Properties dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette. Click **OK**.



Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from the list of those available. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Title	Specifies the benchmark title value
Title Text Max Width	Enter the maximum width of the title text. The title text wraps and adds as many lines as necessary to fit the complete text within the specified width. A control point on the text box allows the width of existing text to be changed manually.
Elevation Display	Select the method for calculating the benchmark's elevation value. • Custom: Allows you to manually input a **Custom Elevation** value • Z value relative to ground plane: Uses the Z value of the elevation benchmark marker, plus the elevation of the object's layer, to determine the elevation value • Z value relative to reference elevation: Uses the Z value of the elevation benchmark marker, plus the elevation of the object's layer, minus the **Reference Elevation** value, to determine the elevation value • Y value relative to reference elevation: Uses the Y value of the elevation benchmark marker, minus the **Reference Elevation** value, to determine the elevation value • Distance from control point: Allows the elevation value to be referenced from a control point displayed on the drawing, rather than from the drawing's active layer plane. If the control point is moved, the elevation value remains set by the distance from the control point to the elevation benchmark object. This allows you to set a control point at the ground floor of a house, for example, so that the elevation benchmark value is set according to the height from the ground floor.
Custom Elevation	If **Elevation Display** is set to Custom, specifies the benchmark elevation value
Reference Elevation	If **Elevation Display** is set to Z Value Relative to Reference Elevation or Y Value Relative to Reference Elevation, specifies the reference elevation value
Style	Select the marker style 
Line Position	For ISO style, specifies the position of the line relative to the marker

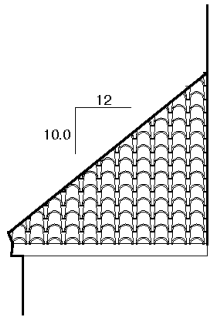
Parameter	Description
Title Position	Specifies the position of the title text relative to the line and marker
Elevation Position	Specifies the position of the elevation value relative to the line and marker
Marker Orientation	Select the marker orientation (left or right)
Marker Graphics	Select a marker style from the list; styles include filled, unfilled, and for ISO style half-filled versions. For the half-filled style, the left half is filled and the right half is unfilled.
2D Scale Factor	Specifies the elevation benchmark size; increase the scale value to obtain a larger object
Crosshair Scale Factor	For US style, increases the size of the elevation benchmark crosshairs, extending them beyond the marker circle;. For ISO style, when the **Line Position** is Top of Marker, draws a horizontal line at the bottom of the marker; a scale factor of 0 draws no line, and a scale factor of 1 makes the line the same width as the marker. 
Marker Size	For ISO style, specifies the marker size; increase the size value to obtain a larger marker
Use Offset	Add an offset to the elevation benchmark's leader line (available only on the Object Info palette after the elevation benchmark object is placed)  For ISO style when the **Line position** is set to Top of Marker, a different type of offset can be created by increasing the **2D Scale Factor** and decreasing the **Marker Size**.
Offset	Specify the size of the offset. A control point on the offset start point allows the offset to be moved horizontally to change the angle of the offset line. It cannot be moved horizontally past the shoulder point.
Shoulder Length	Specify the shoulder length. A control point on the shoulder point allows the shoulder length to be changed manually. The control point can be moved horizontally to adjust the shoulder length; the offset start point moves with it to maintain the angle of the offset line. The control point can also be moved vertically to adjust the offset; the marker moves with it.
Extension Length	Specifies the length of the object line leading to the offset start point (available only on the Object Info palette after the elevation benchmark object is placed) Only the extension length is drawn with the benchmark object's line style set in the Attributes palette. The line portions extending from the offset start point to the marker remain solid.

After creation, multiple elevation benchmark objects can be aligned and distributed with the **Align/Distribute Leader Lines** command; see "Aligning and Distributing Leader Lines" on page 1035.

Using Notation Objects

A L Indicating the Slope of Surfaces

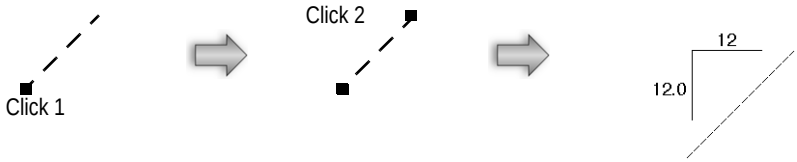
The **Slope Dimension** tool creates a rise over run indicator that displays the slope (rise/run) of any angle on any surface in the drawing.



To create a slope indicator:

1. Click the **Slope Dimension** tool from the Dims/Notes tool set.
2. Click to mark the beginning of the slope. Click again to mark the end of the slope.

   If this is the first time the tool is used in this session, the Slope Dimension Properties dialog box opens. Specify the preferences to use for this tool during this session, and then click **OK**. The preferences can be changed later in the Object Info palette.
3. The rise/run indicator is drawn.



Use the control point at the “elbow” of the slope indicator to move it; change the appearance of the rise/run indicator by editing its parameters in the Object Info palette.

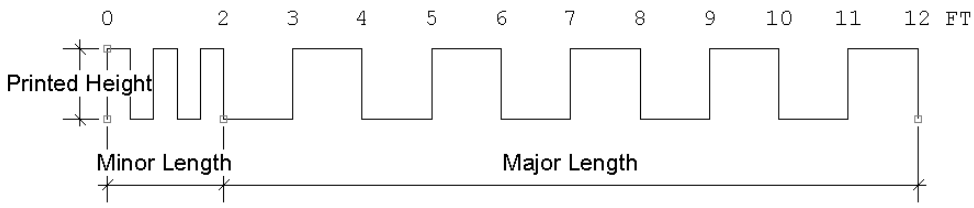
Click to show/hide the parameters.

Parameter	Description
Draw Roof Surface	Draws the defining line as a dashed line
Scale Factor	Determines the size of the indicator
Base	Specifies a numerical denominator to indicate the “run” portion of the rise/run ratio

Using Notation Objects

Inserting a Scale Bar

A scale bar can be placed on the drawing to define the scale of the drawing objects. To include a scale bar for a viewport, select **Modify > Edit Viewport**, and place the scale bar as a viewport annotation.



To add a scale bar to the drawing:

1. Click the **Scale Bar** tool from the Dims/Notes tool set.
2. Click on the drawing to insert the scale bar.
3. Click again to determine the rotation of the scale bar and to place it on the drawing.
4. Position the scale bar as desired. The scale bar properties can be edited later by selecting the scale bar and changing the properties in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Printed Height	Specify the printed height of the scale bar
Minor Length	Specify the minor length
# of Min Divs	Specify the number of minor divisions (the first division is split into minor divisions; the remainder of the scale bar is in major divisions)
Major Length	Specify the major length
# of Major Divs	Specify the number of major divisions
Style	Select the scale bar style: Zigzag, Box, Line, Checker, or Flip
Adjust Units	Select to switch between units specified for the drawing file (**File > Document Settings > Units**); for example, switch between feet and inches
Decimal Points	Select to display units with up to four decimal places

Using Notation Objects

Creating Grid Bubbles

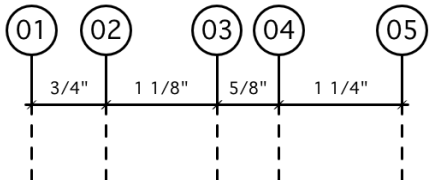
Grid bubble objects are dimensioned grid lines with bubble markers. This type of drawing notation can help locate columns and other primary building features on construction documents.

To draw a grid bubble object:

1. Click the **Grid Bubble** tool from the Dims/Notes tool set.

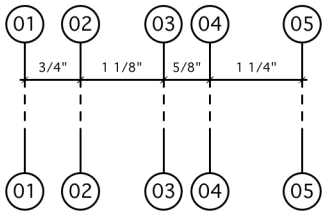
   Place a line of grid bubbles by drawing an open polygon with the tool. The first click starts the grid line; the second click determines the angle of the grid line and sets a marker at that location. Each remaining click sets a grid marker. Grid marker objects maintain a constant size regardless of the drawing scale.

2. Double-click to end the grid line. The first time the object is inserted in this session, the Preferences dialog box opens. These parameters apply to subsequent placements of the object and can be changed in the Object Info palette.

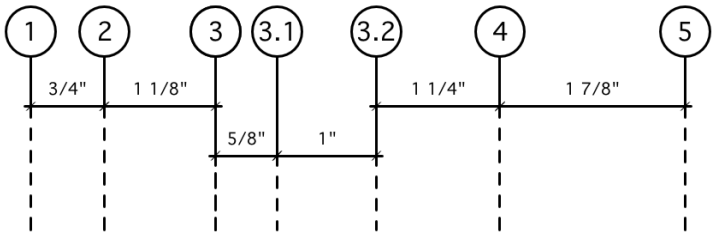


The grid bubble parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Starting Mark	Indicates the starting number or letter for grid labeling; labels are automatically incremented
Prefix	Specifies a prefix, if any, to apply to the grid labels
Dim Text Size	Specifies the point size of the dimension text
Item Text Size	Specifies the point size of the bubble label text
Show Dimensions	Select to display the dimensions on the grid
Flip	Select to flip the bubble labels to the other side of the dimension string
Rotate Text	Select to rotate the bubble label text with the grid
Draw First Grid	Select to display the first bubble grid; deselect to hide it (useful when creating nested grids)
Bubbles Both Ends	Select to draw bubble labels at both ends of the grid line 
Line Style	Select the grid line style
Size Factor	Scales the size of the bubbles
Bubble Offset	Specifies the offset distance of the bubbles from the dimension string
Grid Length	Indicates the grid length
Vertex parameters	Edits the bubble grid line; while editing the bubble grid line in this way is possible, it may be preferable to use the **Reshape** tool instead. See "Reshaping Objects" on page 1043.

By combining several bubble grids and adjusting labeling properties, hierarchical (nested) grids can be created.



Using Notation Objects

Inserting a Symmetry Label

In some drawing standards, when drawing a building or object that is in complete symmetry, the designer may draw only half of the model and insert a symmetry label to indicate the symmetry line.

To insert a symmetry label:

1. Click the **Symmetry Label** tool from the Dims/Notes tool set.
2. In a 2D view, click on the drawing at the desired location to place the symmetry label.
3. Click to set the length and rotation of the label.

   Symmetry label objects are specified and drawn in page scale; they draw to the same apparent size, regardless of the layer scale. If the layer the symmetry label object is on is re-scaled, or the symmetry label object is cut and pasted between layers of different scales, the symmetry label object automatically re-scales.

   The symmetry label's pen color, line weight, and line style can be changed in the Attributes palette.

Using Notation Objects

Error/Revision Management Using Redlines

The **Redline** tool and redline commands help control errors and revisions by providing the ability to annotate drawings with redlines and sketches, and to keep track of redline objects and any changes, corrections, and revisions that occur in a drawing.

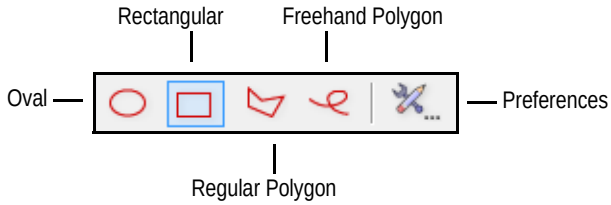
The **Redline** tool and redline commands are compatible with redlines drawn in previous versions and will correctly handle both types of redlines.

Placing Redlines
Attaching a Sketch to a Redline
Show or Hide Redlines
Pick up Redline
Restore Redline
Redline Status Worksheet

Placing Redlines

Use the **Redline** tool to create redline objects, which are graphical time-stamped change requests on the drawing.

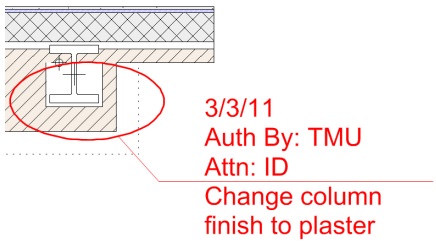
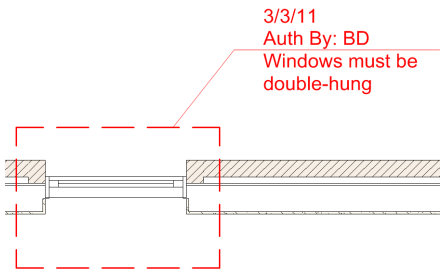
Several modes are available.



To redline an object or area:

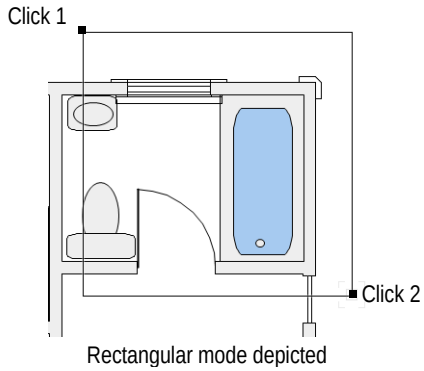
1. Click the **Redline** tool from the Dims/Notes tool set, and select the desired drawing mode from the Tool bar.
2. Click **Preferences** from the Tool bar to specify the **Redline** tool parameters for this session.
   For information on drawing freehand, ovals, polygons, and rectangles, see “Creating Shapes” on page 277.
3. Click **OK**. Draw the redline around the area to be revised or corrected.

Click to show/hide the parameters.

Parameter	Description
Redline Mode	Creates a redline object using an oval, rectangle, polygon, or freehand drawing  Example of oval redline mode
Redline Style	Select the style to apply to the redline object • Cloud - applies a cloud style • Radius - applies corner smoothing with a solid line style • Radius Dashed - applies corner smoothing with a dashed line style • Simple - applies a simple solid line with no smoothing • Simple Dashed - applies a simple dashed line with no smoothing  Example of simple dashed redline style
Authorized by	Specifies the default name or initials of the person who authorized the change; this name is used for the redlines created in this file
Assigned to	Specifies the default name or initials of the person who has been assigned the task; this name is used for the redlines created in this file

Based on the selected creation method, the appropriate tool creates the redline. This allows the use of SmartCursor cues, object snapping, and boomerang mode when drawing redlines.

Draw redlines on the design layer where the error occurred or where the desired revision is to be performed.



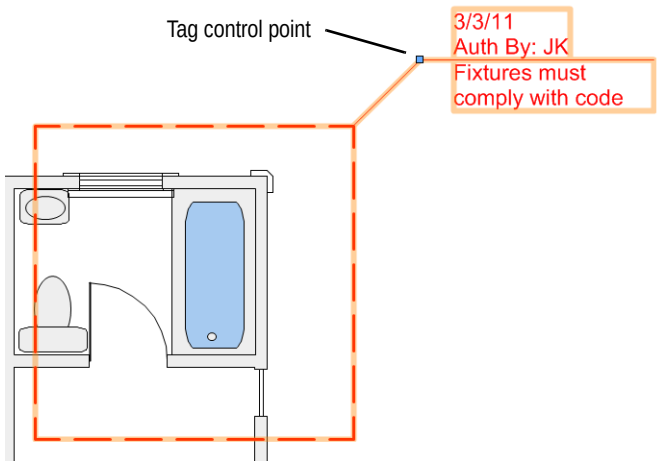
4. The Place Redline dialog box opens. Enter the redline information and authorization.

Click to show/hide the parameters.

Parameter	Description
Redline Notation	Enter text describing the redline condition (and/or specifying the recommended action)
Authorized By	Specifies the name or initials of the person who authorized the change; this originates from the redline preferences and is retained with the file
Assigned To	Specifies the name or initials of the person who has been assigned the task; this originates from the redline preferences and is retained with the file
Attach the current selection as a sketch	Attaches any current selected item(s) to the redline as a symbol to provide more information about the redline (see “Attaching a Sketch to a Redline” on page 1260 for more information)

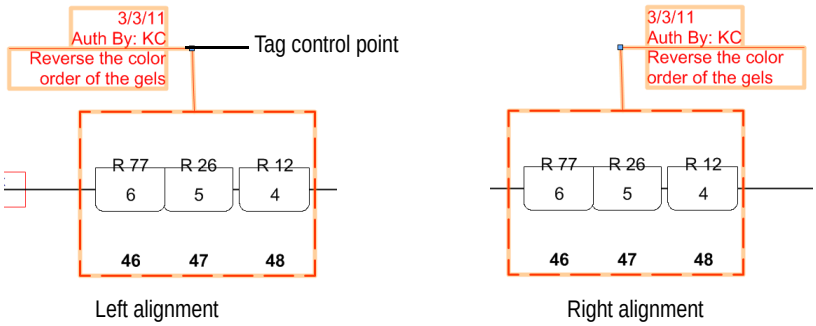
5. Click **OK**.

The redline object is drawn as specified, surrounding the drawing condition to be corrected. The date of the redline is automatically provided. Position redline text by clicking on the tag control point and dragging the tag to the desired location. If there are several redline objects, use the **Align/Distribute Leader Lines** command to improve readability (see “Aligning and Distributing Leader Lines” on page 1035).



The parameters of one or more selected redline objects can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Redline Style	Select a redline style from the list
Tag Alignment	Specifies the alignment of the redline tag: • Auto: Justifies the text away from the center of the redline object • Left: Places the text to the left of the tag control point and right-justifies the text • Right: Places the text to the right of the tag control point and left-justifies the text 
Picked Up	Selecting **Picked Up** indicates that the redline has been resolved or "picked up." The redline color changes to yellow and the pick-up date is automatically assigned. Deselect to indicate that a previously resolved redline has been restored.
Show Authorizer	Select to display the **Authorized By** information on the redline
Authorized By	Displays the initials or name of the person authorizing the change; can be edited
Show Assignee	Select to display the **Assigned To** information on the redline
Assigned To	Displays the initials or name of the person assigned to correct the condition; can be edited
Show Sketch	Select to display the sketch attached to the redline (if any)
Creation Date	Displays the redline creation date
Pick Up Date	Displays the redline pick-up date
Notation	Redline comment information is displayed and can be changed
Tag Length	Controls the length of the line separating the redline comments from the redline information, and changes the redline comment text wrapping

Error/Revision Management Using Redlines
Aligning and Distributing Leader Lines

Attaching a Sketch to a Redline

In screen plane mode, one or more selected objects can be attached to a redline object to illustrate the redline comments. The object to be attached as a sketch must be selected before drawing the redline; this process converts the selected object to a sketch symbol and places it in a Redline Sketches symbol folder saved with the file.

To attach a selected object as a redline sketch:

1. Switch to the screen plane (see “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152).
2. Select the object to be converted to a sketch.
3. Click the **Redline** tool from the Dims/Notes tool set, and draw the redline as described in “Placing Redlines” on page 1257.

   The Place Redline dialog box opens.
4. Select **Attach the current selection as a sketch** and click **OK**.
5. The selection is converted to a symbol that is attached to and moves with the redline object.

   The sketch can be hidden by deselecting **Show Sketch** in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Error/Revision Management Using Redlines
Placing Redlines

Show or Hide Redlines

Over the course of a project, most drawings receive a large number of redlines, which, if left visible, would clutter the drawing. Select **Text > Redlines > Show or Hide Redlines** to toggle redline visibility.

All Redline objects are drawn in the Redlines class. If redlines are hidden when a new redline is drawn, the Redlines class visibility is automatically turned on and all the redlines become visible. Use the **Show or Hide Redlines** command to hide them again.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Error/Revision Management Using Redlines

Pick up Redline

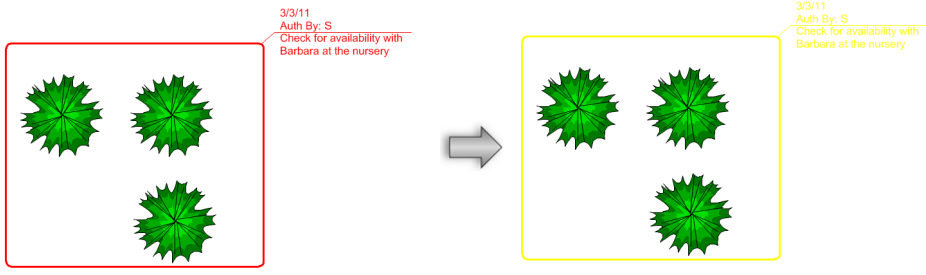
Once the change or correction indicated by the redline has been resolved, the redline needs to be “picked up” or changed to a closed state.

To pick up a redline:

1. Select the redline object or objects.
2. Select **Text > Redlines > Pick Up Redline**.

   Right-click on the redline and select **Pick Up Redline** from the context menu.

   This sets the status of all selected redline objects to closed. The redline color changes from red to yellow, and the pick-up date is set.



Not picked up Picked up

Redlines can be picked up by selecting one or more redline objects and selecting **Picked Up** in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Error/Revision Management Using Redlines

Restore Redline

A selected closed redline object can be restored to an open status. A redline object may need to be restored when a revision has not been performed satisfactorily.

To change the state of a closed redline back to open:

1. Select the redline object or objects.
2. Select **Text > Redlines > Restore Redline**.

   Right-click on the redline and select **Restore Redline** from the context menu.

   The selected redlines are returned to an open status and the redline color changes back to red. The original redline creation date is displayed. Redlines can also be restored by selecting one or more redline objects and deselecting **Picked Up** in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Error/Revision Management Using Redlines

Redline Status Worksheet

The redline worksheet lists the status of all redlines in the current file. The Open Redlines section lists all redlines that require resolution; the Closed Redlines section shows all resolved redlines. For each redline item, the design layer location, date, and redline comment displays.

To create or open the redline status worksheet:

1. Select **Text > Redlines > Create Redline Status WS**.
2. The Redlines Status Report worksheet is created and opens automatically. It is added to the **Window > Worksheets** menu and it is listed in the Resource Browser.

   Redlines, both open and closed, are listed in the report even if they are hidden in the current view of the file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Error/Revision Management Using Redlines

Using the ID Label Tool

The **ID Label** tool labels drawing objects with an identifier so they can be properly referenced on an associated schedule. These labels can be applied to specific objects with attached data records, such as doors, windows, equipment, plumbing fixtures, electrical items, and irrigation objects. This tool also provides a convenient way to specify or edit attached record data. The ID labels can be customized using special symbols (see “Creating Custom ID Label Symbols” on page 1266 for more information).

For doors and windows, the ID label bubble size is independent of text size. The ID bubble is drawn at the size specified in the **ID Label Size** field of the Object Info palette, unless the text exceeds the ID bubble size; in this case, the bubble automatically expands to fit the text. To restore the ID bubble to a fixed size, reduce the text size to fit within the ID bubble.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To place an ID label:

1. Click the **ID Label** tool from the Dims/Notes tool set.
2. Click to set the location of the ID label.
3. Click on the object to associate with the ID label.

   The ID/Specification Tool dialog box opens. Set the ID parameters.
4. Click **Options** to set specific preferences for this ID.

Click to show/hide the parameters.

Parameter	Description
ID Type	Select the record associated with the ID; this is automatically selected if the object is associated with one of the available records
ID Style	Select the graphical ID style to display If a styled wall is being labeled, this is the only field available; data fields must be added and edited via the Resource Browser, as described in "Editing ID Labels and Records for Wall Styles" on page 1266.
Duplicate Existing ID	Click to associate an existing ID to this label; select the ID from the list
Create New ID	Click to create a new ID label
Prefix	Assigns alphanumeric information before the numerical label value; adding prefix information is optional
Label	Assigns a numerical value to the ID; this number increments automatically if the auto-increment option is chosen in the Set ID Preferences dialog box
Suffix	Assigns alphanumeric information after the label value if the auto-increment option is chosen; adding suffix information is optional
Select a field to specify	Select a record field associated with the object to edit its value
Selected field value	Enter or edit a field value and then click **Set** to confirm the change
Options	For windows or doors, opens the ID Settings dialog box; for other objects, opens the Set ID Preferences dialog box

If the object being labeled is a window or door, complete the fields in the ID Settings dialog box as described in the ID Tag tab section of "Window Settings: ID Tag Pane" on page 597 and "Door Settings: ID Tag Pane" on page 620. Select **Auto-Increment ID Label** in the ID Settings dialog box to automatically increment the numerical ID value each time the label is placed. Continue with step 6. For all other objects, go to step 5.

5. If the object being labeled is something other than a window or door, complete the fields in the Set ID Preferences dialog box.

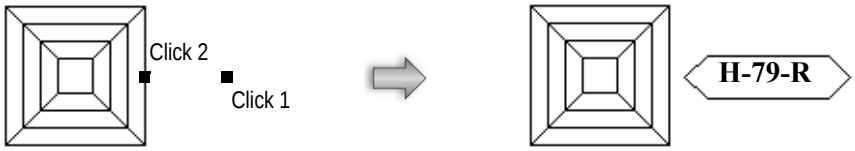
Click to show/hide the parameters.

Parameter	Description
Auto-Increment ID Label	Automatically increments the numerical ID value each time the label is placed; not available for styled walls

Parameter	Description
Fix current record when matching IDs	Automatically overwrites the current record information when applying an existing ID label; not available for styled walls
Write ID fields of records with matchfield	Finds records with matching value in match field and writes ID value to match; not available for styled walls
Draw Leader from ID to object	Draws a leader from the ID label to the object
Rotate ID with Leader	Rotates the ID symbol to the angle of the leader line
Length of leader shoulder	Sets the length of the horizontal segment of the leader line
Arrow style	Sets the style of arrow to use with the leader line
Marker scale factor	Sets the size of the leader's arrow; a factor of less than one decreases the size of the arrow while a factor of more than one increases its size

6. Click **OK**.

   The ID label is drawn.



To bypass the ID Label dialog boxes and automatically place an existing ID label repeatedly, hold down the Alt (Windows) or Option (Mac) key while applying additional labels.

To easily convert any object into a door or window, place an ID label on the object and select the Window Object or Door Object **ID Type** in the ID/Specification Tool dialog box. A symbol is automatically created for the object and displays in the Resource Browser, and is available to use as symbol geometry for subsequently placed windows or doors. Select the window or door object and click **Settings** in the Object Info palette to open the Window or Door Settings dialog box to attach record information to include in the window or door Schedule (see "Inserting Windows in Vectorworks Design Series" on page 594 and "Inserting Doors in Vectorworks Fundamentals" on page 609).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Existing ID Labels and Record Information
Vectorworks Architect and Landmark ID Symbols

A L Editing Existing ID Labels and Record Information

After placement, objects with record information and ID labels can be edited to change the record or ID data. Worksheets, including schedules, and the **ID Label** tool can change the record information of multiple objects at once.

Editing Record Information with Worksheets

Global changes to symbol and plug-in object record information can be made using database rows in worksheets created with the Vectorworks program. During a schedule review, it is often necessary to make changes to objects and update them automatically on the drawing.

Only database rows are directly associated with drawing elements, and can update the drawing in this way. Fields which result from a calculation, or from locked objects, workgroup-referenced items, or control points, cannot edit drawing elements. For more information on worksheets, see "Using Worksheets" on page 1319.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To edit record information from a worksheet:

1. Open the worksheet that contains the record information to be edited, by selecting the worksheet in the Resource Browser and selecting **Edit** from the **Resources** menu or the context menu. Alternatively, select the worksheet from the **Window > Worksheets** menu.

   For example, a Window Schedule worksheet contains record information for the windows in the drawing.

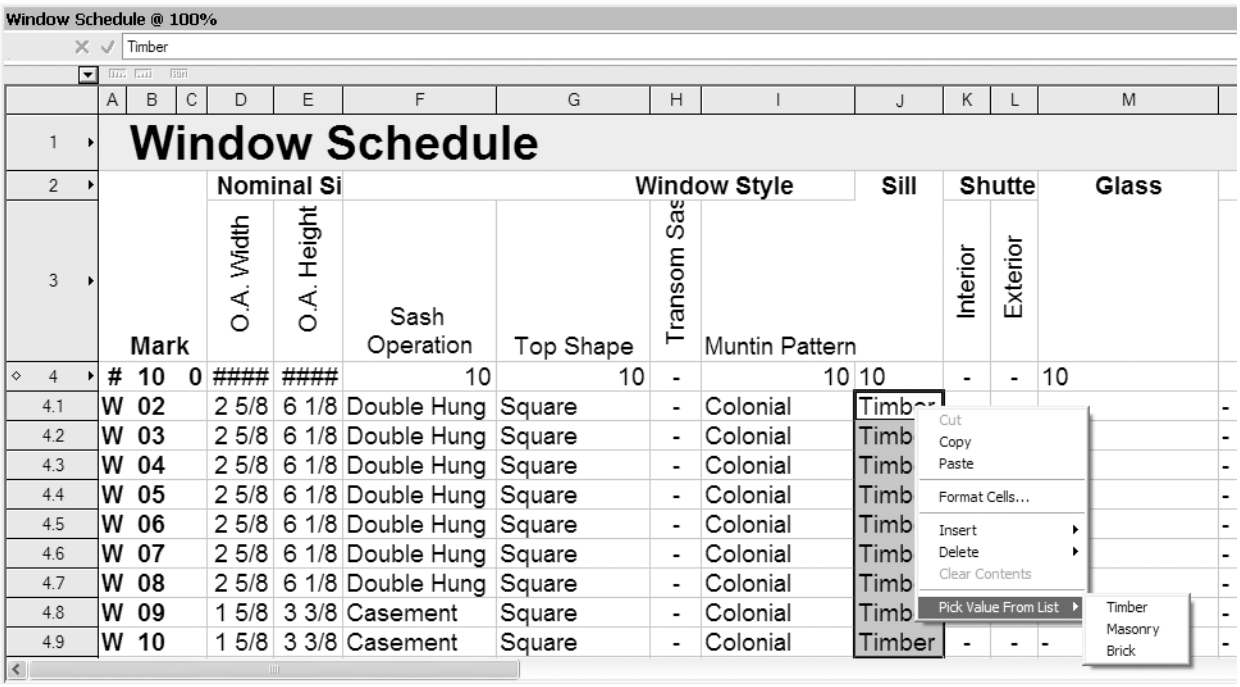
2. Select the database row cell to be edited.

   Text and number fields can be edited directly, while popup fields allow a selection to be made, either from the edit list at the top of the worksheet, or from the **Pick Value from List** context menu item.

   To repeat the same value in several text or number fields of the same record, copy the desired value from one field, select multiple rows, and paste. (Pasting cannot be performed for popup fields.)

3. The drawing records automatically update with the new information, and the drawing objects reflect the changes.

   If record field from a summarized database row has been edited, all objects referenced by the row are edited.



Editing ID or Record Information with the ID Label Tool

The **ID Label** tool can be used to change ID or record information after placing an ID. Labels and records for styled walls are edited differently; see “Editing ID Labels and Records for Wall Styles” on page 1266 for details.

To edit an existing ID label or record information:

1. Click the **ID Label** tool from the Dims/Notes tool set.
2. Double-click on the ID label to edit.

   If the labeled object is a window or door, complete the fields as described in the Data tab section of “Window Settings: Data Pane” on page 605 and “Door Settings: Data Pane” on page 629. If editing the object through the Object Info palette, an additional parameter, **ID Label Size**, is available for controlling the size of the ID label bubble. Continue with step 4. For all other objects, go to step 3.

3. If the labeled object is something other than a window or door, edit the fields in the ID/Specification Tool dialog box, as described in “Using the ID Label Tool” on page 1262.
4. Click **OK**.

   The edits are applied to the ID label and record attached to the object.

Editing ID Labels and Records for Wall Styles

Data fields (including an ID label) are specified for each wall style, which means that each wall with a particular style has the same label and record attached to it. Therefore, ID labels and records cannot be edited for individual styled walls with the **ID Label** tool. (See “Using Wall Styles” on page 507 for more information about creating, editing, and using wall styles.)

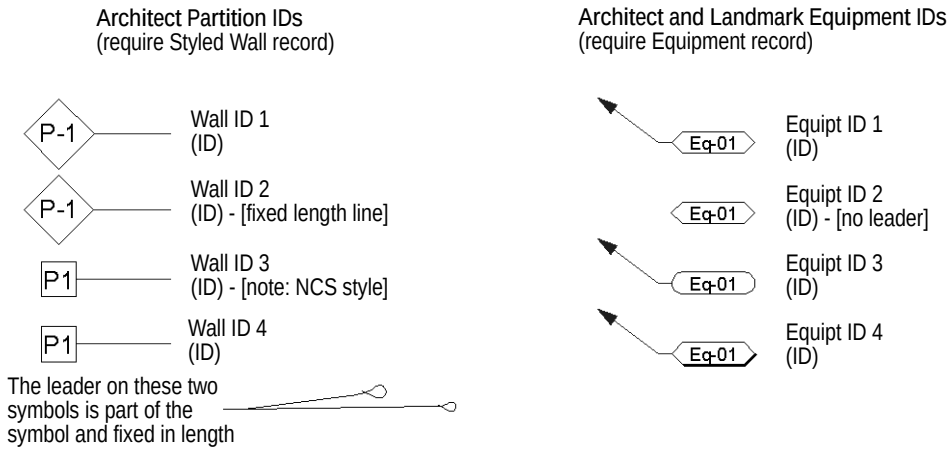
To edit ID label or record information for a wall style:

1. Select the wall style from the Resource Browser and click **Resources > Edit**.
2. In the Edit Wall Style dialog box, click the Data Fields tab.
3. Enter a label ID in the **Mark** field, and enter other data fields as necessary.
4. Click **OK** to close the Edit Wall Style dialog box, and click **OK** again to close the Wall Replacement dialog box.

   From this point forward, walls that are created with this style have the specified data record attached; when an ID label is added to a wall that has this style, the **Mark** entry displays in the label.

A L Vectorworks Architect and Landmark ID Symbols

The following ID symbols are provided with the Vectorworks Architect and Landmark products, and require the presence of the indicated record format in the drawing file. The **ID Label** tool creates the record format if it does not already exist.



Sheet Border Properties

A L Creating Custom ID Label Symbols

Understanding ID Labels

The Vectorworks Architect and Landmark ID labels are created as plug-in objects that draw an ID symbol and an optional leader line. The symbols can automatically be kept horizontal, or can be rotated to the leader line angle.

It is possible to create ID symbols based on custom markers. ID symbols have certain requirements that must be met.

ID Symbol Requirements

The **ID Label** tool (see “Using the ID Label Tool” on page 1262) detects the presence of any existing record information for the object being labeled in the drawing; if the tool cannot detect record information, you are prompted to select the appropriate record from a list. The ID symbols defined to work with that record type display for selection.

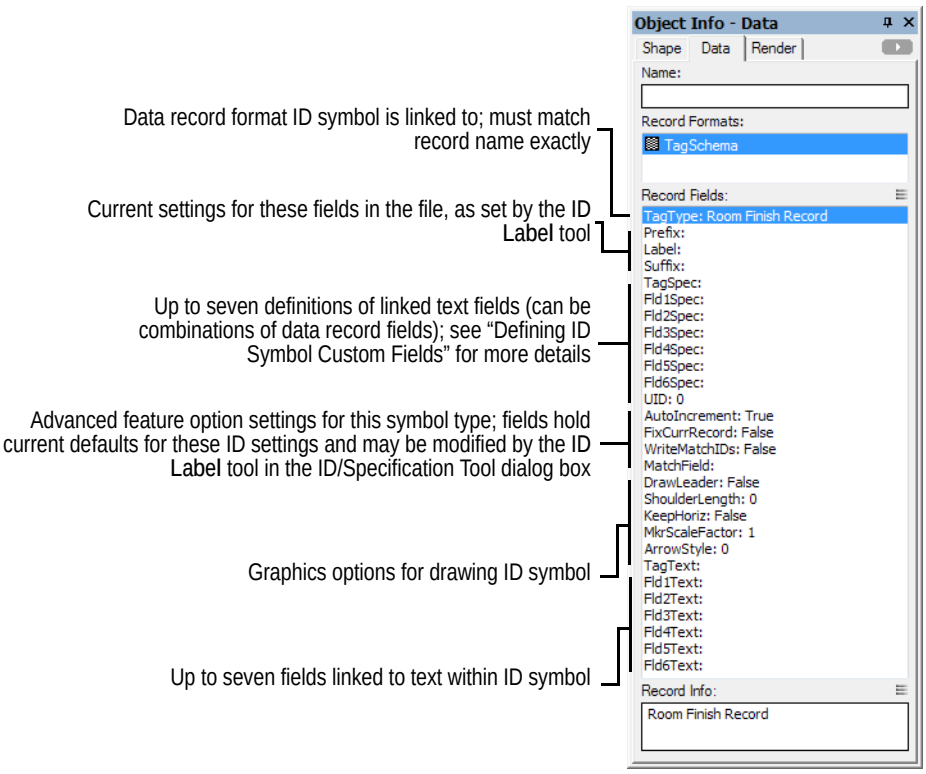
The **ID Label** tool determines whether the ID symbol requested in the dialog box is present in the drawing. If not, it copies it in from the ID_Symbols.vwx file, and scales it to the current drawing layer scale. Once an appropriate scaled symbol is created, it is used without further reference to external documents.

ID symbols must meet the following requirements:

- They are stored in the library file [Vectorworks]\Plug-Ins\Common\Data\ID_Symbols.vwx;
- They are created at a 1:1 scale, using certain graphic primitives;
- They use linked text to display ID and attribute information; and,
- They have a TagSchema record attached.

The TagSchema Record

The TagSchema record determines the behavior of the ID symbol text. This record is present in the ID_Symbols.vwx file.

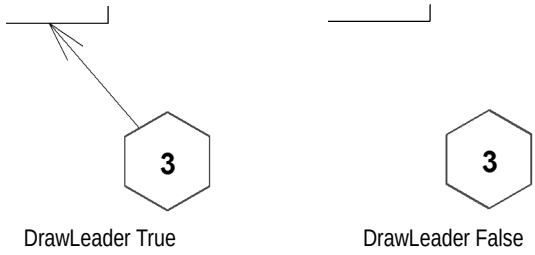
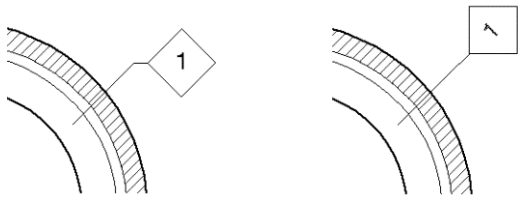
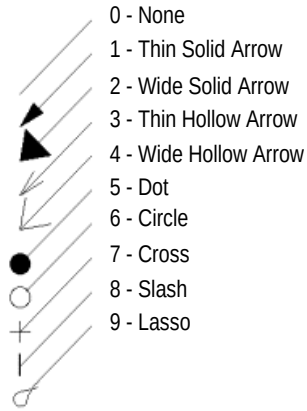


The **AutoIncrement**, **FixCurrRecord**, **WriteMatchIDs**, **DrawLeader**, and **KeepHoriz** fields are Boolean (true/false) values; **ShoulderLength** and **MkrScaleFactor** are numerical values. All other fields are text values.

ID Symbol Leader Options

Four TagSchema record fields determine whether the ID symbol is drawn with a leader, and, if so, determine the leader’s appearance.

Click to show/hide the parameters.

Parameter	Description
DrawLeader	If **DrawLeader** is false, no leader is drawn with the ID Leader object; otherwise, a leader is drawn between the ID symbol and the object to which the ID is referring. This field sets the default for the draw leader parameter in the ID Leader object, which can be changed later.  DrawLeader True DrawLeader False
KeepHoriz	If **KeepHoriz** is false, the ID symbol is drawn rotated to the leader angle; otherwise, the symbol is rotated horizontally, with a small "shoulder" line segment which is controlled using the value in the **ShoulderLength** field. These fields set the defaults for the **KeepHoriz** value and the **ShoulderLength** value of the ID Leader object, respectively. IDs with no leader should normally set **KeepHoriz** to true.  KeepHoriz True KeepHoriz False
ArrowStyle	This field, along with **MkrScaleFactor**, controls the appearance of the marker at the end of the leader (if any) 
MkrScaleFactor	This field, along with **ArrowStyle**, controls the appearance of the marker at the end of the leader (if any); this setting is a multiplier on a marker size of approximately 1/4". For example, a **MkrScaleFactor** setting of 0.5 results in a marker of 1/8" standard size.

ID Symbol Record Writing Options

There are three ID record writing options controlled by four fields in the TagSchema record:

Click to show/hide the parameters.

Parameter	Description
AutoIncrement	The value in this field controls the default setting for whether the ID number is automatically incremented between placements
FixCurrRecord	The value in this field controls whether the contents of the current record are rewritten to match earlier IDs with the same ID entry
WriteMatchIDs	The value in this field controls the writing of matching ID fields to records with identical contents in the **MatchField** text field
MatchField	Provides a record field for writing the ID for matching

Defining ID Symbol Custom Fields

The ID symbol can have up to seven fields containing virtually any text, combined with the contents of the data record the ID uses.

For example, the **ID Label** tool reads the contents of the **Fld1Spec** field and writes the results to the **Fld1Text** field. The "Spec" field follows the format **FieldName& "string constant"** where the field names are fields in the data record named in the **TagType** field; the & indicates a concatenation, and the string constants are surrounded by double quotes.

For example, a field in the data record is called "Count." This is a number field representing the total number of something. You want the first field in the ID to read: **TOTAL: 12** when "Count" is 12. Enter the following formula in the **Tag1Spec** field: **"TOTAL: "&Count**. There are no spaces between the ampersand and the field name, or between the string constant (in quotes) and the ampersand.

You can concatenate any number of fields and constants. For example, you could add the word "Item(s)" to the tag field definition by using the following formula: **"TOTAL: "&Count& " Item(s)"**.

There are a variety of pre-defined fields in the ID symbols that come with the Vectorworks Architect and Landmark products. Use these as a guideline in understanding this special formula language. This same formula convention is also used to define HVAC object tags.

Error Messages in ID Fields

If the **ID Label** tool cannot find the data record specified in the **TagType** field of the TagSchema record, the error message **#RECORDNAME?#** displays when the ID is placed. If any of the fields in the formula are misnamed, the message **#FIELDNAME?#** displays in the affected ID field. Verify the spelling of record and field names and ensure the data record is defined as described in "The Data Record" on page 1269.

The Data Record

The data record named in the **TagType** field of the ID symbol should be part of the standard records created using the **VA Records and Schedules** command and should be present in the current preference set. This enables the **ID Label** tool to create the record automatically if it is not defined at the time the **ID Label** tool is used.

The TagType "Styled Wall" is reserved for use when an ID is placed on a styled wall.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Custom ID Symbols

AL Creating Custom ID Symbols

Keeping in mind the information described in "Understanding ID Labels" on page 1266, it is possible to create a custom ID symbol to be used by the **ID Label** tool.

An ID symbol can consist of any 2D object supported in the Vectorworks program except circles, arcs, grouped objects, or other symbols. These guidelines are necessary for the scaling algorithm currently used by the **ID Label** tool, to scale
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

the symbol to the correct scale for the file. To achieve the look of an arc in the symbol, use an arc-smoothed vertex of a polyline; use an oval instead of a circle.

To create a custom ID symbol:

1. Open the ID Symbols.vwx file, located in the [Vectorworks]\Plug-Ins\Common\Data folder.

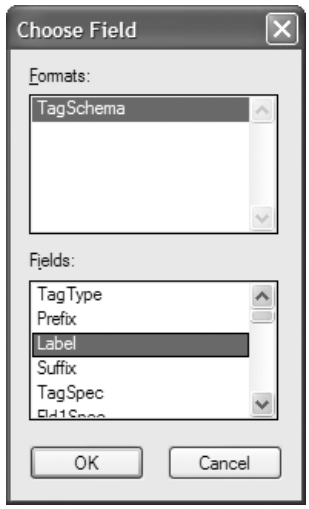
   The TagSchema record, as well as the ID symbols currently used by the **ID Label** tool, are present in this file.
2. At a 1:1 scale, draw the object representing the ID symbol.
3. Select **Modify > Create Symbol** to create a symbol from the object (see “Creating New Symbols” on page 239 for more information on creating symbols).
4. Select the new symbol in the Resource Browser and select **Resources > Edit**.

   The Edit Symbol dialog box opens.
5. Select the 2D Component and click **Edit** to open the Edit Symbol window.
6. Deselect all items by clicking in an empty area.

   On the Data tab of the Object Info palette, *SYMBOL DEFAULTS* is displayed.
7. Attach the Tag Schema record by selecting it from the Data tab.
8. An “X “displays in the check box.
9. Select the TagType field and replace the field name with the exact name of your custom record.
10. Create and format text to be used to display the ID symbol text.

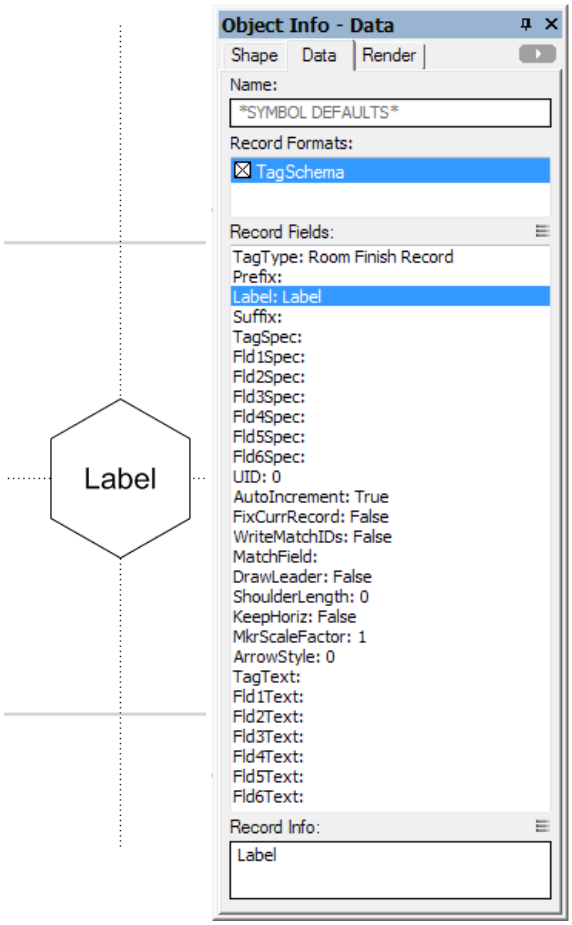
    For more information, see “Linking Text to Record Formats” on page 269.
11. With the text selected, select **Tools > Records > Link Text to Record**.

    The Choose Field dialog box opens.



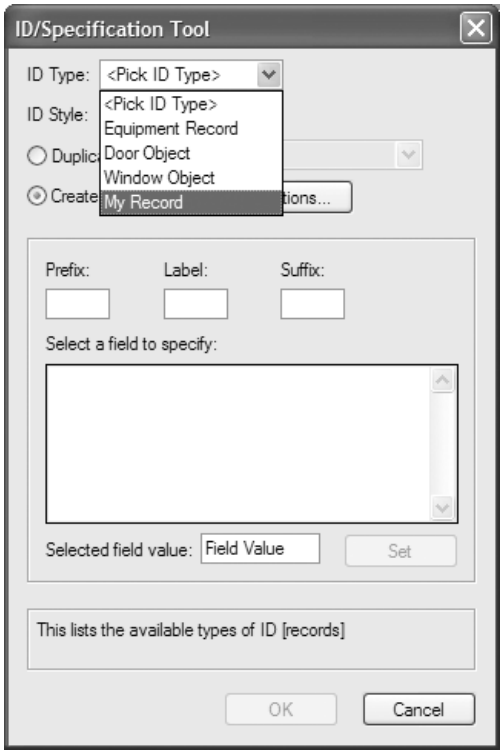
12. In the Tag Schema record format, select the Label field and click **OK**.
13. The symbol text is linked to display the contents of the Label record field.

    To check the text link, enter text in the Label field on the Data tab of the Object Info palette. Data entered now would be overwritten later, in any case, by the **ID Label** tool.



14. Click **Exit Symbol** at the top right corner of the window to return to the drawing.
15. In the Pally script palette, double-click the **Output ID Prefs** script to run it.
16. Save the ID_Symbols.vwx file.
17. In the drawing file where the custom ID symbol is to be used by the **ID Label** tool, ensure that a record format exists which exactly matches the record name entered previously in the TagType field.
18. Select the **ID Label** tool from the Dims/Notes tool set.

    The ID/Specification Tool dialog box opens.



19. The custom record is one of the selections for **ID Type**. Select it to use the associated custom ID symbol and click **OK**.

D Section Note

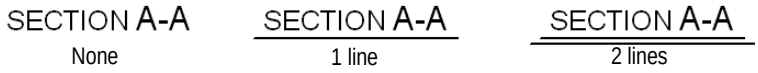
The **Section Note** object annotates the drawing or detail with section letters, and, if desired, a scale. The text attributes are editable by the **Format Text** command.

To insert a section note:

1. Click the **Section Note** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Section	Specifies the section note
% of Title	Specifies the size of the section letters or numbers in relation to the section name. Enter a percentage between 50 to 200.
Scale (opt)	Displays the scale information below the section information, if desired SECTION A-A SCALE: 2:1

Parameter	Description
% of Title	Specifies the size of the scale name and number in relation to the section name. Enter a percentage between 50 to 200.
All Caps	Select to display the section and scale names in capital letters
Underline	Select the underline style, if desired 

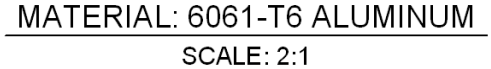
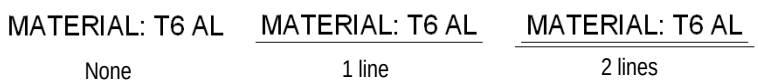
Material Note

The **Material Note** object annotates the drawing or detail with material composition, and, if desired, scale. The text attributes are editable by the **Format Text** command.

To insert a material note:

1. Click the **Material Note** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Material	Specifies the material note
% of Title	Specifies the size of the material letters or numbers in relation to the material name. Enter a percentage between 50-200.
Scale (opt)	Displays the scale information below the material information, if desired 
% of Title	Specifies the size of the scale name and number in relation to the material name. Enter a percentage between 50 to 200.
All Caps	Select to display the material name in capital letters
Underline	Select the underline style 

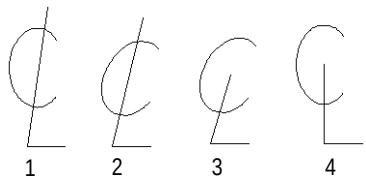
Center Line Marker

The **Center Line Marker** object denotes the center line on drawings.

To insert a center line marker object:

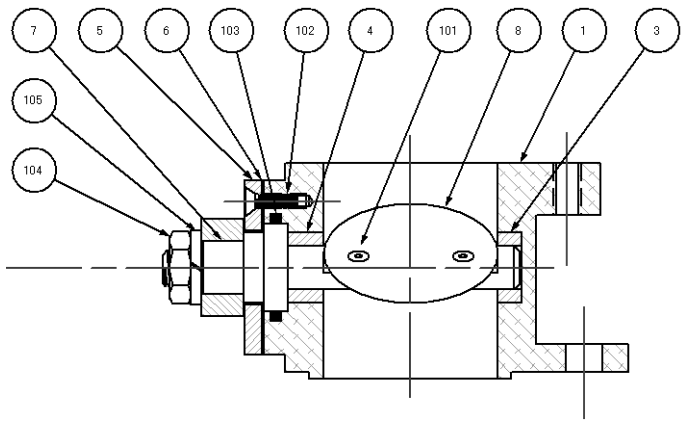
1. Click the **Center Line Marker** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Size (on paper)	Specifies the size of the object
Style	Select the center line marker style 

Creating Detail Bubbles

A detail bubble assigns a detail number to the parts of an assembly drawing.



The information associated with the detail bubble can be used to create both parts lists and bill of materials lists.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Detail Bubbles
Detail Bubble Record Format
Editing Detail Bubbles
Aligning and Distributing Leader Lines

Adding Detail Bubbles

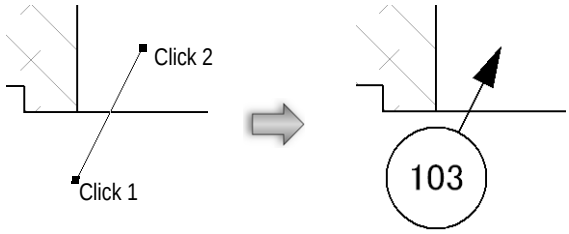
To add a detail bubble:

1. Click the **Detail Bubble** tool from the Dims/Notes tool set.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Click on the drawing to place the bubble portion of the object. Click again to place the marker.

   If this is the first time the object has been inserted in this session, the Object Properties dialog box opens. If the Attributes palette has not been set to use end markers, select the marker style, size, and angle from the Object Properties dialog box.

   If no Part Info record exists, one is created automatically the first time a detail bubble is placed in the drawing.



   The detail bubble number automatically increments as the bubbles are placed. Change the auto-incrementing number by editing the **Item Number** field in the Object Info palette. When detail bubble placement resumes, the last number entered becomes the starting number.

Creating Detail Bubbles

Detail Bubble Record Format

The Part Info record format stores the information for the detail bubble. This record format can be edited to add new fields or to delete existing fields. To edit the record format, right-click on the record format in the Resource Browser and select **Edit**. For more information on record formats, see “Record Formats” on page 262.

If new fields are created, the existing Bill of Materials and Parts List worksheets need to be modified to reflect the changes. Alternatively, create new worksheets to accommodate the new fields. For more information on using worksheets, see “Using Worksheets” on page 1319.

Creating Detail Bubbles

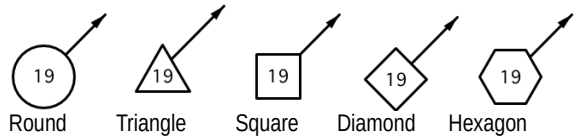
Editing Detail Bubbles

Detail Bubble Properties

Edit the detail bubble properties in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file’s content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See “Using Text Styles” on page 393 and “Formatting Text” on page 389.
Min. Bubble Size	Enter the minimum bubble size

Parameter	Description
Shape	Select one of the bubble shapes 
Item Number	Specify the number to display in the detail bubble; the item number is used for bill of materials and parts list worksheets
Change Bubble Info	Click to edit the bubble detail

Once the detail bubble has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).

Editing Detail Bubble Record Information

To edit the record information associated with a detail bubble:

1. Select the bubble, and then click **Change Bubble Info** on the Object Info palette.

   The Edit Detail Bubble dialog box opens.

2. Enter the information associated with the detail bubble. The information can be used to complete the parts list and bill of materials worksheets.

3. Click **OK** to return to the Object Info palette. If the **Item #** was changed, the change is automatically made to the Object Info palette and the object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Detail Bubbles
Marker Attributes

Creating a Bill of Materials

The information associated with the detail bubbles can be included in a worksheet which automatically generates a bill of materials list.

To create a bill of materials:

1. Once the detail bubble record information has been completed, select the **Create Bill of Materials** command from the appropriate menu:

   - Architect workspace: **AEC > Machine Design > Create Bill of Materials**
   - Landmark workspace: **Landmark > Machine Design > Create Bill of Materials**
   - Spotlight workspace: **Spotlight > Machine Design > Create Bill of Materials**

2. With the bull's-eye cursor, click in the drawing to place the worksheet.

   Alternatively, access the Bill of Materials worksheet object with the Resource Browser from the title blocks file in the [Vectorworks]\Libraries folder (see "Resource Libraries" on page 219). Double-click the Bill of Materials worksheet object in the Resource Browser to place the worksheet graphic object.

3. Double-click on the worksheet, or select **Window > Worksheets > Bill of Materials**, to open the worksheet. From the **Worksheet** menu, select **Recalculate**. The record information from the detail bubbles automatically fills the worksheet, and all calculations are performed. For more information on worksheets, see "Using Worksheets" on page 1319.

4. Close the worksheet. The worksheet graphic object on the drawing reflects the changes.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

5. Reopen the worksheet and select **Recalculate** from the **Worksheet** menu to update the worksheet when any changes are made to the record information.

   A parts list can be added to an ASME title block; see “Sheet Border Properties” on page 78.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Parts List
Creating Detail Bubbles
Creating Reports

Creating a Parts List

The information associated with the detail bubbles can be included in a worksheet that automatically generates a parts list.

To create a parts list:

1. Once the detail bubble record information has been completed, select the **Create Parts List** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Create Parts List**
   - Landmark workspace: **Landmark > Machine Design > Create Parts List**
   - Spotlight workspace: **Spotlight > Machine Design > Create Parts List**
2. With the bull’s-eye cursor, click in the drawing to place a Parts List worksheet.

   Alternatively, select the appropriate parts list worksheet object with the Resource Browser from the title blocks file in the [Vectorworks]\Libraries folder (see “Resource Libraries” on page 219). Double-click the appropriate parts list worksheet object in the Resource Browser to place the worksheet graphic object.

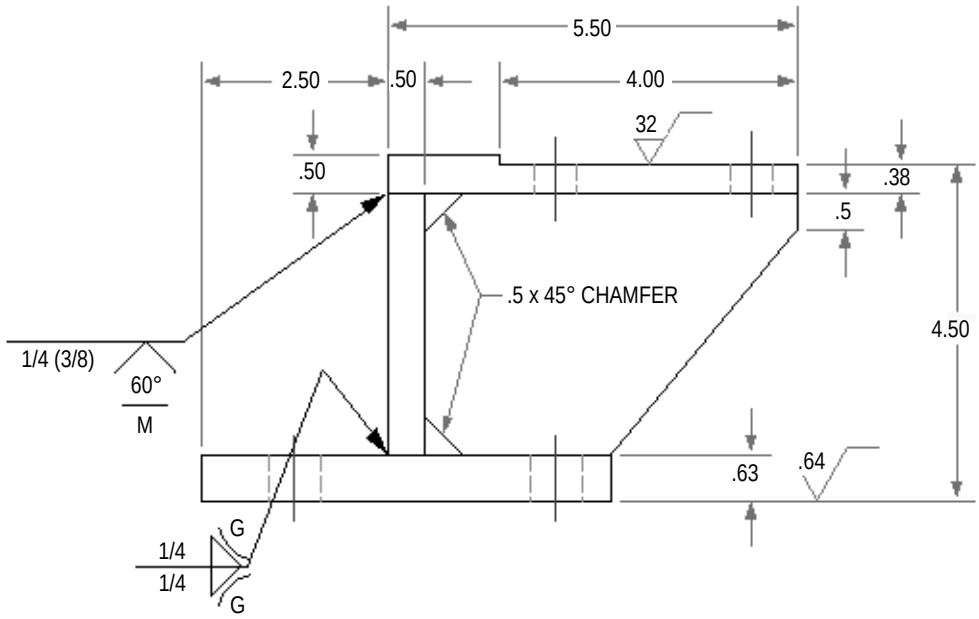
   The Parts List-1 worksheet object is formatted to fit the ASME title blocks for ASME A, B, and C and ISO A2, A3, A4, and A5 drawing sizes. The Parts List-2 worksheet object is formatted to fit the ASME title blocks for ASME D, E, and F, and ISO A0 and A1 drawing sizes.
3. Double-click on the worksheet, or select **Window > Worksheets > Parts List** to open the worksheet. From the **Worksheet** menu, select **Recalculate**. The record information from the detail bubbles automatically fills the worksheet, and all calculations are performed. For more information on worksheets, see “Using Worksheets” on page 1319.
4. Close the worksheet. The worksheet graphic object on the drawing reflects the changes.
5. Reopen the worksheet and select **Recalculate** from the **Worksheet** menu to update the worksheet when any changes are made to the record information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Bill of Materials
Creating Detail Bubbles
Creating Reports

Welding and Surface Texture Symbols

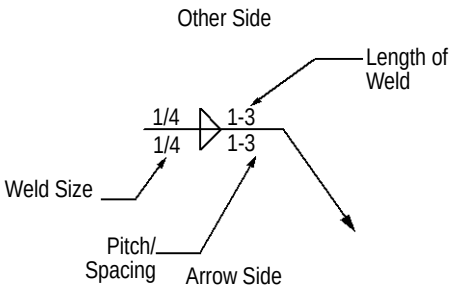
Vectorworks Design Series products contain a comprehensive collection of welding symbols and also include a surface texture symbol. All symbols are placed as point objects.



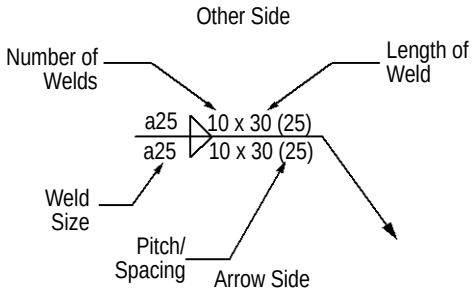
Fillet Welding Symbol

To insert a fillet welding symbol:

1. Click the **Welding Sym-Fillet** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.
4. Once the object has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).

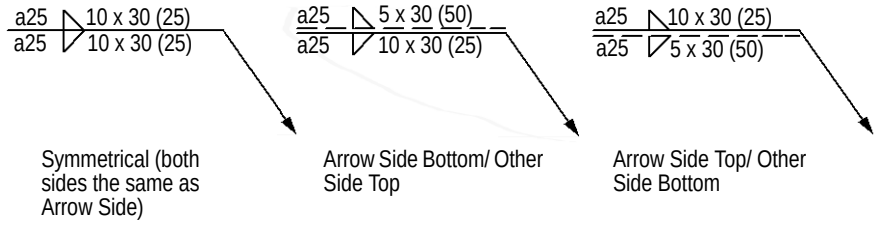
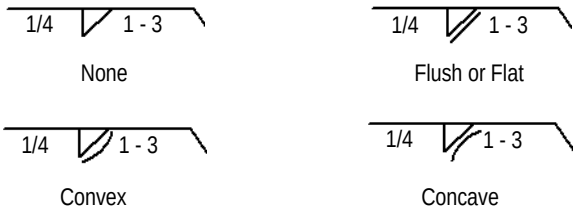


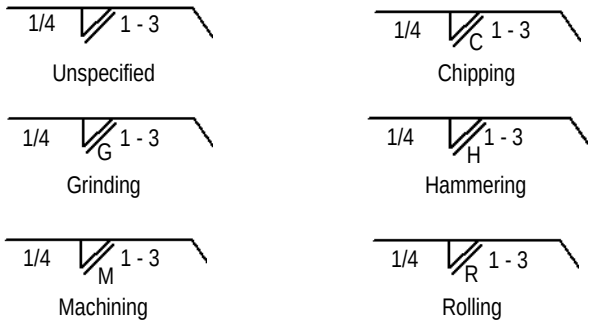
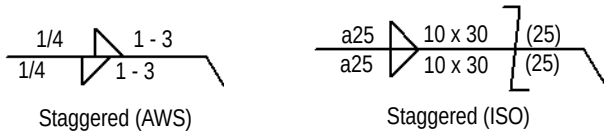
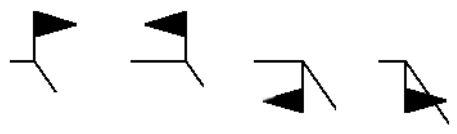
Welding Symbol - Fillet (AWS)

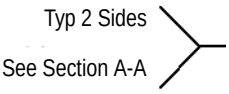


Welding Symbol - Fillet (ISO)

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Symmetrical (both sides the same as Arrow Side) (AWS Only)	Select to reflect the same information on both the arrow side and the other side
Configuration (ISO Only)	Select the data display configuration; if symmetrical is selected, the data on the other side is identical to the arrow side  Symmetrical (both sides the same as Arrow Side) Arrow Side Bottom/ Other Side Top Arrow Side Top/ Other Side Bottom
Arrow Side/Other Side	Select to display data on the arrow side, other side or both; if symmetrical is selected, the displayed data applies to both sides
Weld Size	Displays the size of the weld
Number of Welds (ISO Only)	Displays the number of welds
Length of Weld	Displays the length of the weld
Pitch/Spacing	Displays the pitch or spacing of the weld
Finish Contour	Select the finish contour type  None Flush or Flat Convex Concave

Parameter	Description
Finish Method	Select the finish method type. A finish contour must be selected in order to display a finish method.  Finish methods are shown with the flat or flush contour
Stagger	Select to indicate staggered welds 
Arrow	Sets whether to have an arrow on the leader line
Position	Select the side from which the arrow on the leader line extends (left or right)
Reference	Specify whether to include a reference note
Position	Select the position of the reference note; by default, **Left** is selected if the arrow is on the right, or **Right** is selected if the arrow is on the left
Ref. Line 1	If desired, enter the data to display in the first reference line
Ref. Line 2	If desired, enter the data to display in the second reference line
Ref. Line 3	If desired, enter the data to display in the third reference line
Weld All Around	Select to place a weld all around the marker
Field Weld	Select to place a field weld flag
Position	Sets the position of the field weld flag. Select **Top** |> to have the flag on top facing right, **Top** <| to have the flag on top facing left, **Bottom** |> to have the flag on bottom facing right, **Bottom** <| to have the flag on bottom facing left. 

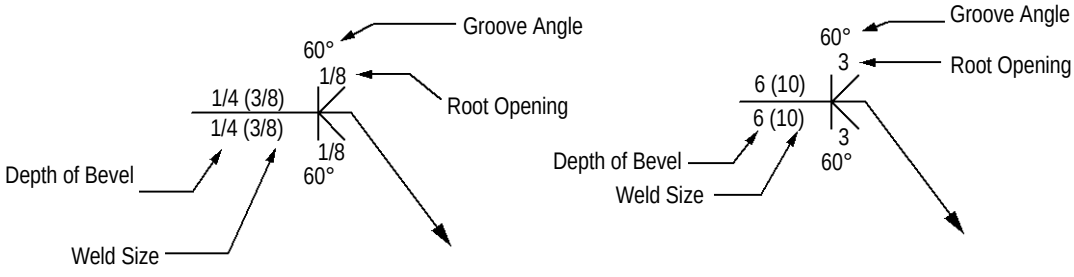


Marker Attributes

D Groove Welding Symbol

To insert a groove welding symbol:

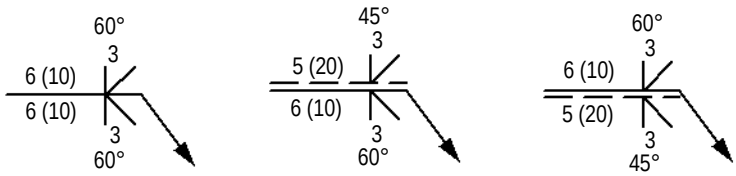
1. Click the **Welding Sym-Groove** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.
4. Once the object has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).

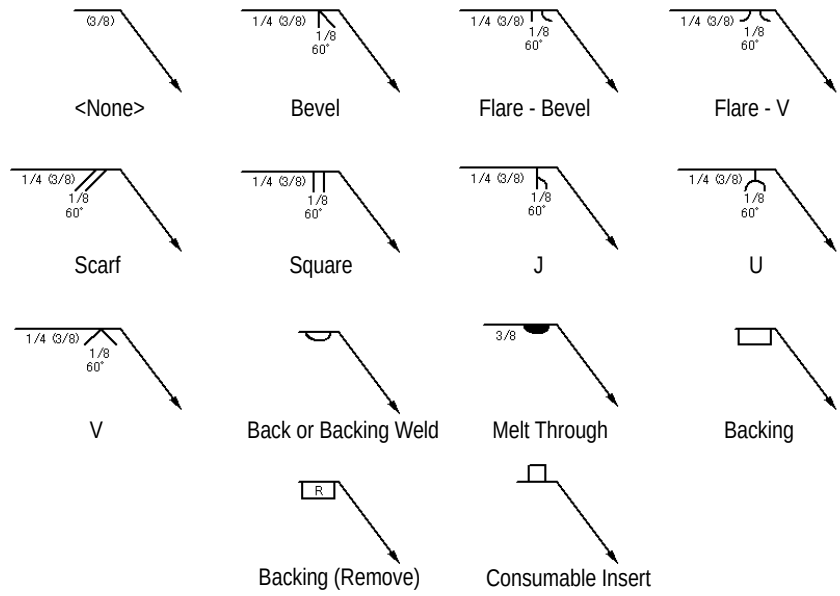
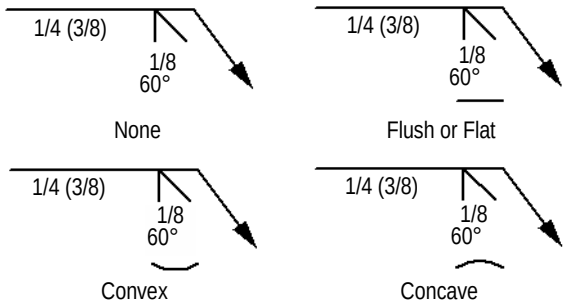
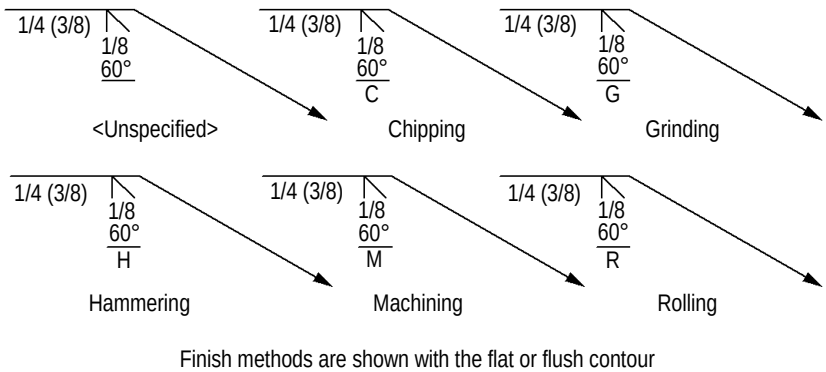


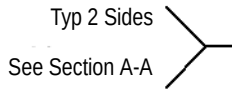
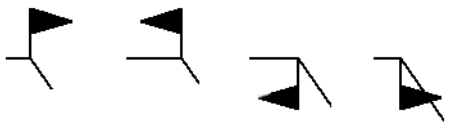
Welding Symbol - Groove (AWS)

Welding Symbol - Groove (ISO)

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Symmetrical (both sides the same as Arrow Side) (AWS Only)	Select to reflect the same information on both the arrow side and the other side
Configuration (ISO Only)	Select the data display configuration; if symmetrical is selected, the information on the other side is identical to the arrow side  Symmetrical (both sides the same as Arrow Side) Arrow Side Bottom/ Other Side Top Arrow Side Top/ Other Side Bottom
Arrow Side/Other Side	Select to display data on the arrow side, other side or both; if symmetrical is selected, the displayed data applies to both sides

Parameter	Description
Weld Symbol	Select the type of groove weld symbol to be displayed 
Weld Size	Displays the size of the weld
Depth of Bevel	Displays the depth of the bevel
Root Opening	Displays the root opening
Groove Angle	Displays the groove angle
Finish Contour	Select the finish contour type 
Finish Method	Select the finish method type 

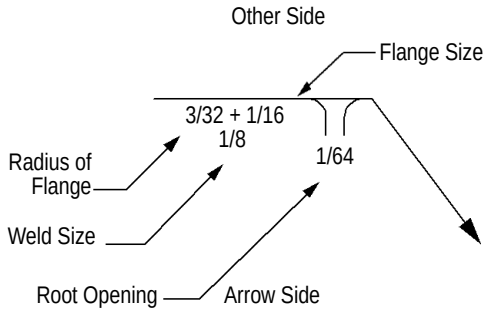
Parameter	Description	
Arrow	Sets whether to have an arrow on the leader line	
Position	Select the side from which the arrow on the leader line extends (left or right)	
Reference	Specify whether to include a reference note	
Position	Select the position of the reference note; by default, **Left** is selected if the arrow is on the right, or **Right** is selected if the arrow is on the left	
Ref. Line 1	If needed, enter the data to display in the first reference line	
Ref. Line 2	If needed, enter the data to display in the second reference line	
Ref. Line 3	If needed, enter the data to display in the third reference line	
Weld All Around	Select to place a weld all around the marker	
Field Weld	Select to place a field weld flag	
Position	Sets the position of the field weld flag. Select **Top |>** to have the flag on top facing right, **Top <|** to have the flag on top facing left, **Bottom |>** to have the flag on bottom facing right, **Bottom <|** to have the flag on bottom facing left. 	

Welding and Surface Texture Symbols
Editing the Arrow Segment
Marker Attributes

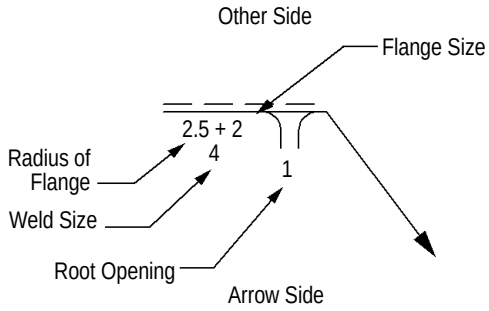
D Flange Welding Symbol

To insert a flange welding symbol:

1. Click the **Welding Sym-Flange** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.
4. Once the object has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).

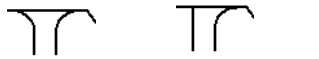
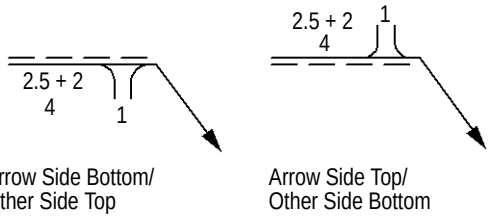
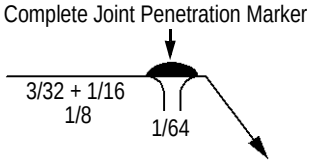


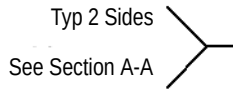
Welding Symbol - Flange (AWS)



Welding Symbol - Flange (ISO)

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Type of Weld	Sets the weld type 
Configuration (ISO Only)	Select the data display configuration 
Significance	Select to display data on the arrow side or the other side
Weld Size	Displays the size of the weld
Flange Size	Displays the size of the flange
Radius of Flange	Displays the radius of the flange
Root Opening	Displays the size of the root opening
Complete Joint Penetration	Select to display a complete joint penetration marker 
Arrow	Sets whether to have an arrow on the leader line
Position	Select the side from which the arrow on the leader line extends (left or right)

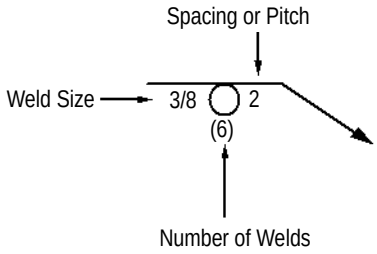
Parameter	Description	
Reference	Specify whether to include a reference note	
Position	Select the position of the reference note; by default, **Left** is selected if the arrow is on the right, or **Right** is selected if the arrow is on the left	
Ref. Line 1	If needed, enter the data to display in the first reference line	
Ref. Line 2	If needed, enter the data to display in the second reference line	
Ref. Line 3	If needed, enter the data to display in the third reference line	

Welding and Surface Texture Symbols
Editing the Arrow Segment
Marker Attributes

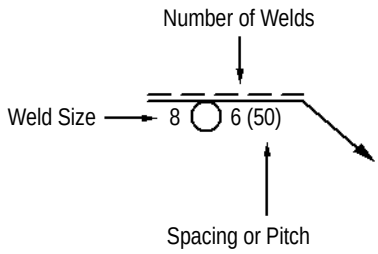
D Miscellaneous Welding Symbols

To insert a miscellaneous welding symbol:

1. Click the **Welding Sym-Misc** tool from the Dims/Notes tool set to insert a spot, seam, stud, or surfacing welding symbol.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.
4. Once the object has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).



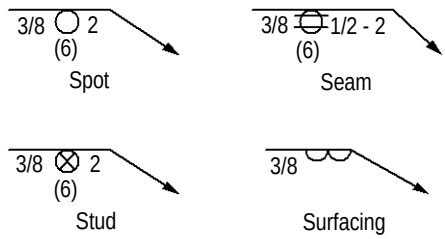
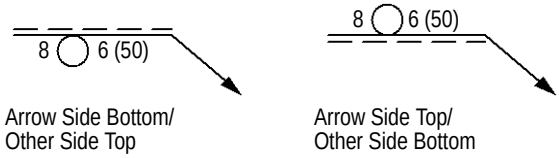
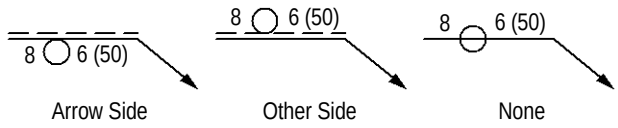
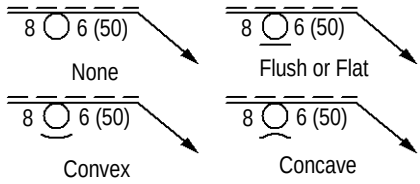
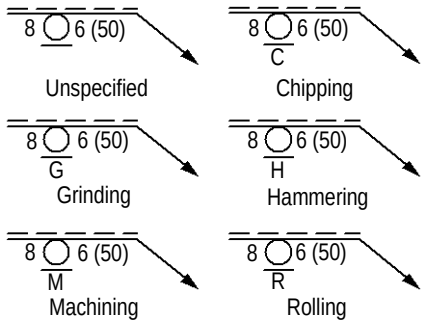
Welding Symbol - Miscellaneous (AWS) Spot Weld

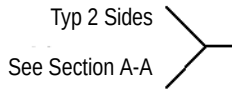
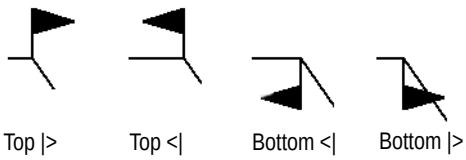


Welding Symbol - Miscellaneous (ISO) Spot Weld

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)

Parameter	Description
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Type of Weld	Select the weld symbol type 
Configuration (ISO Only)	Select the data display configuration 
Significance	Sets where data displays about the leader line 
Weld Size	Displays the size of the weld. Applies to all weld types.
Length	Displays the length of the weld. Applies to the seam weld only.
Spacing or Pitch	Displays the desired spacing or pitch. Applies to spot, seam, and stud welds only.
Number of Welds	Displays the number of welds. Applies to spot, seam, and stud welds only.
Finish Contour	Select the finish contour type 
Finish Method	Select the finish method type 

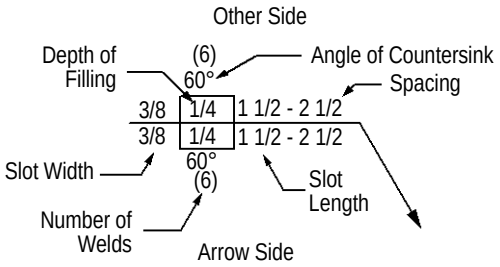
Parameter	Description	
Arrow	Sets whether to have an arrow on the leader line	
Position	Select the side from which the arrow on the leader line extends (left or right)	
Reference	Specify whether to include a reference note	
Position	Select the position of the reference note; by default, **Left** is selected if the arrow is on the right, or **Right** is selected if the arrow is on the left	
Ref. Line 1	If needed, enter the data to display in the first reference line	
Ref. Line 2	If needed, enter the data to display in the second reference line	
Ref. Line 3	If needed, enter the data to display in the third reference line	
Weld All Around	Select to place a weld all around the marker	
Field Weld	Select to place a field weld flag	
Position	Sets the position of the field weld flag. Select **Top** |> to have the flag on top facing right, **Top** <| to have the flag on top facing left, **Bottom** |> to have the flag on bottom facing right, **Bottom** <| to have the flag on bottom facing left. 	

Welding and Surface Texture Symbols
Editing the Arrow Segment
Marker Attributes

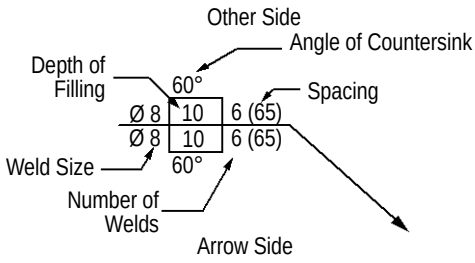
D Slot-Plug Welding Symbol

To insert a slot-plug welding symbol:

1. Click the **Welding Sym-Slot-Plug** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.
4. Once the object has been placed, its marker can be selected from the Attributes palette (see "Marker Attributes" on page 1102).

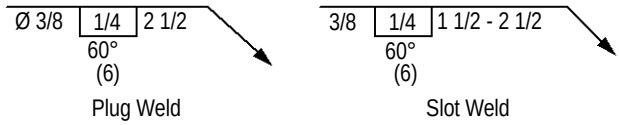
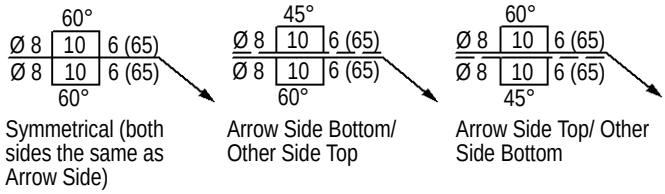


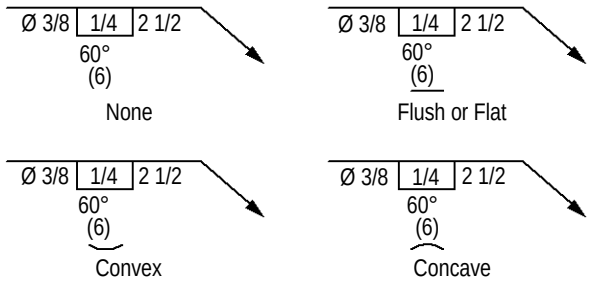
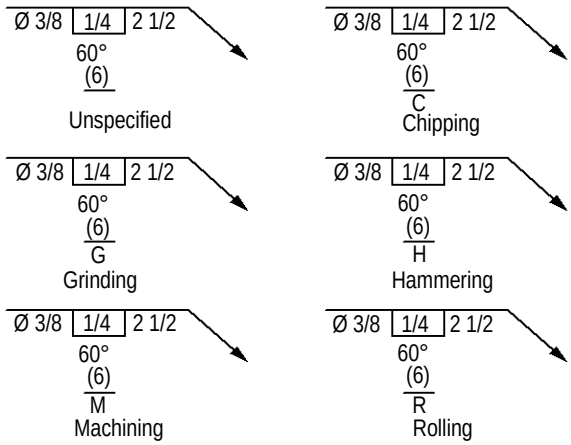
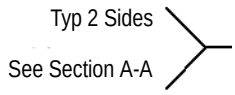
Welding Symbol Slot-Plug (AWS) Slot Weld

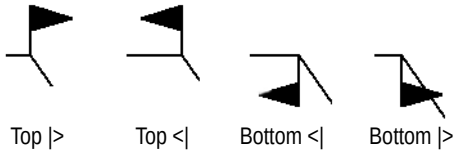


Welding Symbol Slot-Plug (ISO) Plug Weld

Click to show/hide the parameters.

Parameter	Description
Rotation	Specifies the number of degrees to rotate the object (0.00 is horizontal)
Text Style	Select a text style from either the default content or the current file's content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>. See "Using Text Styles" on page 393 and "Formatting Text" on page 389.
Type of Weld	Select the weld type 
Symmetrical (both sides the same as Arrow Side) (AWS Only)	Select to reflect the same information on both the arrow side and the other side
Configuration (ISO Only)	Select the data display configuration; if symmetrical is selected, the information on the other side is identical to the arrow side 
Arrow Side/Other Side	Select to display data on the arrow side, other side or both; if symmetrical is selected, the displayed data applies to both sides
Weld Size (Plug Weld Only)	Displays the weld size
Slot Width (Slot Weld Only)	Displays the width of the slot
Slot Length (Slot Weld Only)	Displays the length of the slot
Angle of Countersink	Displays the angle of the countersink
Depth of Filling	Displays the depth of the filling
Spacing	Displays the spacing
Number of Welds	Displays the number of welds

Parameter	Description	
Finish Contour	Select the finish contour type 	
Finish Method	Select the finish method type 	
Arrow	Sets whether to have an arrow on the leader line	
Position	Select the side from which the arrow on the leader line extends (left or right)	
Reference	Specify whether to include a reference note	
Position	Select the position of the reference note; by default, **Left** is selected if the arrow is on the right, or **Right** is selected if the arrow is on the left	
Ref. Line 1	If needed, enter the data to display in the first reference line	
Ref. Line 2	If needed, enter the data to display in the second reference line	
Ref. Line 3	If needed, enter the data to display in the third reference line	
Weld All Around	Select to place a weld all around the marker	
Field Weld	Select to place a field weld flag	

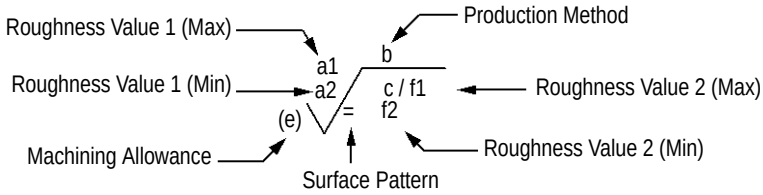
Parameter	Description
Position	Set the position of the field weld flag. Select **Top** |> to have the flag on top facing right, **Top** <| to have the flag on top facing left, **Bottom** |> to have the flag on bottom facing right, **Bottom** <| to have the flag on bottom facing left. 

Welding and Surface Texture Symbols
Editing the Arrow Segment
Marker Attributes

D Surface Texture Symbol

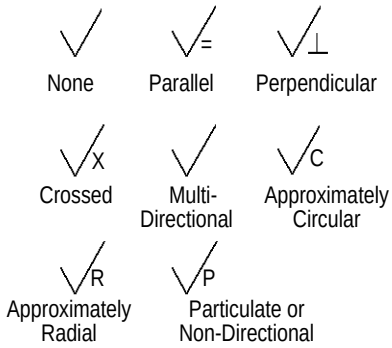
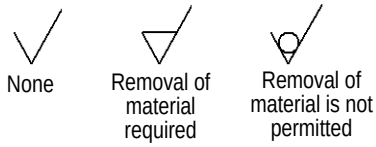
To insert a surface texture symbol:

1. Click the **Surface Texture Symbol** tool from the Dims/Notes tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

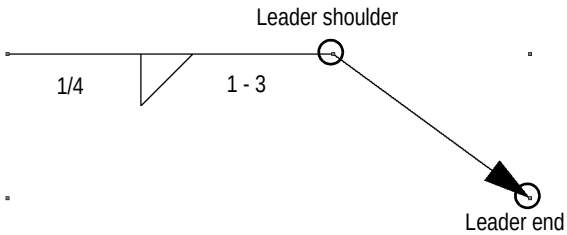
Parameter	Description
Roughness Value 1 (Max)	Displays the primary maximum roughness value
Roughness Value 1 (Min)	Displays the primary minimum roughness value
Waviness Height	Displays the waviness height
Roughness Value 2 (Max)	Displays the secondary maximum roughness value
Roughness Value 2 (Min)	Displays the secondary minimum roughness value

Parameter	Description
Surface Pattern	Select the surface pattern type 
Machining Allowance	Displays the machining allowance
Production Method	Displays the production method
Special Conditions	Select the special condition 
Same Condition on All Surfaces	Select to add a **Same Condition on All Surfaces** marker 
Rotate Text with Symbol	Select to rotate the text with the symbol
Clear All Fields	Select to delete the current information from all fields

Welding and Surface Texture Symbols
Editing the Arrow Segment
Marker Attributes

Editing the Arrow Segment

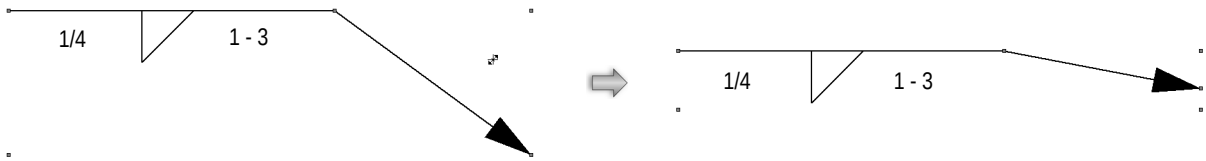
The arrow segment of each welding symbol has two control points for adjusting the segment position.



To move the control points of an arrow segment:

1. Select the welding symbol to edit.
2. Click and hold the leader end control point.

3. Drag the leader end control point to the new location and click to set.



4. Click and hold the leader shoulder control point.
5. Drag the leader shoulder control point to the new location and click to set.

Notes Management

The Vectorworks Design Series products provide a full set of features for managing and creating complete and fully-referenced annotation of project files. These features can streamline the annotation process by providing a central location for storing, selecting, and placing frequently-repeated notes. Notes can be reused across files, either within a project, or across separate projects.

A callout is text with a leader line, displaying information about an item in the drawing; a callout object is included with the Vectorworks Fundamentals product. Its capabilities are extended in the Vectorworks Design Series products, allowing it to be used for keynotes in addition to ordinary notes. A keynote is a callout that references a note contained in a keynote legend. Placing callouts as keynotes also creates a keynote legend object. The keynote legend automatically numbers the notes, and ensures that multiple instances of the same note are assigned the same keynote number. General notes, also available in the Vectorworks Design Series products, contain text not associated with a particular item in the drawing.

The callout, keynote legend, and general notes objects can be used with manually-entered text, or in conjunction with external databases which store and retrieve text. When a database is used, text is entered only once into the database, and then pulled from the database for notes in the drawing. After notes have been placed, if changes are made to the database or to the notes, the differences can be reconciled with the **Reconcile Notes** command.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

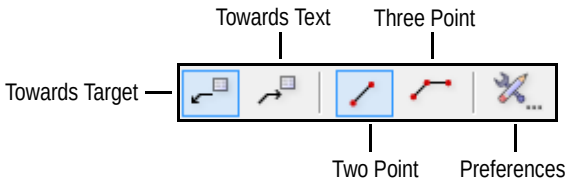
Notes Management without Databases

Inserting Callouts or Keynotes

The **Callout** tool places callout objects on a drawing. A callout object is a block of text attached to a leader line with an optional bubble surrounding the text. Use callout objects to annotate items in a file.

In the Vectorworks Design Series products, the **Callout** tool includes extended capabilities which allow it to be used for keynotes. A keynote legend is automatically created.

A keynote can be used without a database as described in this section, or in conjunction with an external notes database (see “Placing Database Callouts” on page 1304).



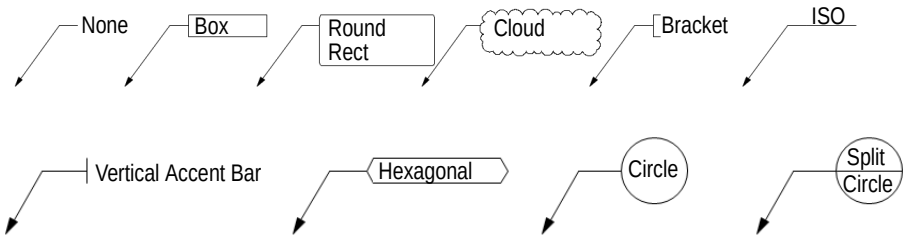
Mode	Description
Towards Target	Click first where the callout text is to be placed, and then near the object to be annotated
Towards Text	Click first near the object to be annotated, and then where the callout text is to be placed
Two Point	Two clicks are required to place the callout object; in Two Point mode, the length of the shoulder is determined in the callout preferences or Object Info palette
Three Point	Three clicks are required to place the callout object; in Three Point mode, the third click determines the length of the shoulder
Preferences	Opens the Callout Preferences dialog box
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To create a callout or (for Vectorworks Design Series products) a keynote:

1. Click the **Callout** tool from the Basic palette.
2. Click the **Preferences** button on the Tool bar. Specify the callout or keynote object preferences, which apply to new callouts created either in this file or all files. These parameters can be changed later for a selected callout object in the Object Info palette. If a marker type is selected in the Preferences, the marker can be changed later from the Attributes palette.

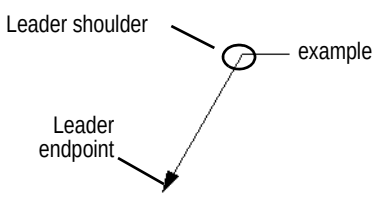
Click to show/hide the parameters.

Parameter	Description
Text Options	
Get Text from Database (Vectorworks Design Series required)	Database controls are not available in the Vectorworks Fundamentals product. In the Vectorworks Design Series products, the **Callout** tool can be used in conjunction with an external notes database; see “Placing Database Callouts” on page 1304 for information on using a notes database.
Place As Keynote (Vectorworks Design Series required)	Creates a keynote with keynote legend. Specify the keynote prefix and suffix text, if desired. Keynote legends are layer-specific; one legend is created per layer. However, a callout placed as a keynote can be associated with a keynote legend on any layer. Select the legend for the callout from the **Keynote Legend** list in the Callout Preferences dialog box or the Object Info palette of a selected keynote. The Default Legend is the keynote legend located on the current layer.
Rotate Text	When selected, rotates the callout text or keynote number to the specified **Text Angle**
Text Angle	Specifies the angle of text rotation; certain angles may not be available, depending on the text’s **Horizontal Position** settings
Max. Text Width	Indicates the maximum text width before text wraps; if the text string is shorter than maximum width, the bubble sizes to fit the text. Rotated text cannot restrict the maximum text width.
Vertical Position	Sets the vertical position of the text relative to the shoulder; select **Auto** to align the top text line to the shoulder if the leader is on the left, or to align the bottom text line to the shoulder if the leader is on the right
Horizontal Position	Sets the horizontal position of the text relative to the shoulder; select **Auto** to position the text to the right if the leader is on the left, or to the left if the leader is on the right
Horizontal Text Alignment	Specify how to align callout text. Horizontal text alignment for existing callouts can also be set by selecting **Text > Alignment** or **Text > Format Text** and selecting the desired alignment option.
Format Text	Opens the Format Text dialog box, to set text attributes or select a text style

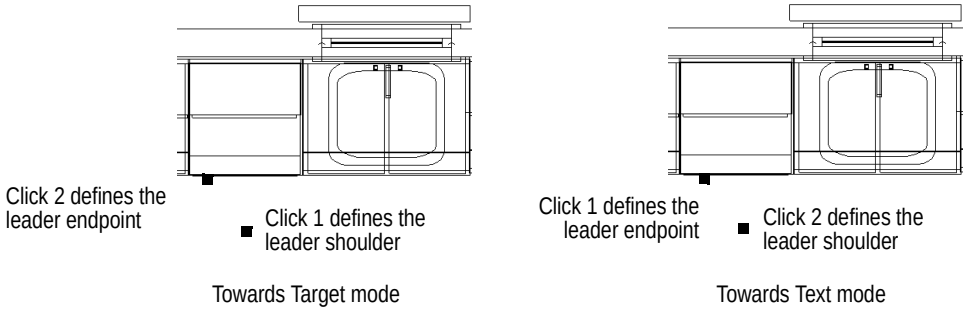
Parameter	Description
Bubble Options	
Bubble Style	Select the type of bubble to draw around the callout text or keynote number 
RR Corner Radius	For Round Rect bubble styles, sets the corner radius
Text Margin	Sets the distance between the bubble and the text
Bubble Shadow	Select to draw the bubble with a drop shadow (does not apply to None, Bracket, ISO, or Vertical Accent Bar styles)
(Bubble) Shadow Settings	If **Bubble Shadow** is selected, opens the Bubble Shadow Settings dialog box, for specifying bubble shadow edge and fill attributes; see "Specifying Callout or Keynote Legend Bubble Shadow Settings" on page 1297
Leader Options	
Shoulder Length	Sets the length of the line between the text and the start of the leader; can be changed by moving a control point or in the Object Info palette. In Three Point mode, this length is set by the third mouse click.
Leader Type	Select Line, Arc, Bézier, or None; curved leader lines contain additional control points for controlling the curve shape
Leader Radius	For arc leader types, specifies the arc radius
Marker	Select a marker style from the marker style list, or select **Custom** to create a custom marker. Select **Edit Marker List** to open the Edit Marker List dialog box; see "Editing the Marker List" on page 1103.
Other Options	
Apply Settings to New Callouts in	Select whether these callout preferences should apply to new callouts or keynotes in this file only or globally, to all future files

3. Click **OK** to set the callout or keynote preferences.
4. Click the desired insertion modes from the Tool bar, and then click in the drawing to select the insertion point of the callout or keynote object.

   Depending on the mode, the first click defines the leader shoulder or the leader endpoint.



5. Click again to determine either the leader endpoint or leader shoulder, depending on the mode.



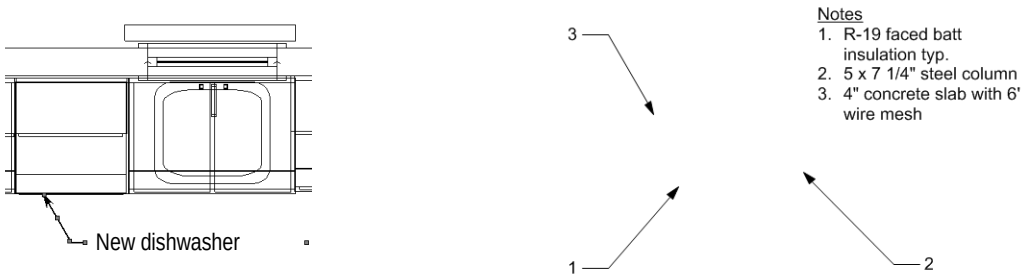
6. If in **Three Point** mode, click a third time to define the shoulder length.

   The Notes Manager: Callout dialog box opens. Enter the callout or keynote legend text; text wraps if longer than the specified maximum text width. Press Enter to add a carriage return.

   To place callout text both above and below the line in the Split Circle **Bubble Style**, press Enter between the lines of text.

7. Click **OK**.

   The callout or keynote is placed in the drawing; if a keynote is placed, a keynote legend object is also created.



Click here for a video tip on this topic (Internet connection required).

Editing a Callout Object
Editing Keynotes and the Keynote Legend
Formatting Text

Editing a Callout Object

Editing Callout Text

To edit the text of an existing callout object:

1. Select the callout object to edit.
2. Either double-click on the callout object with the **Selection** tool, or click **Edit Note** from the Object Info palette.

   The Notes Manager: Callout dialog box opens.

3. Enter the desired text changes.

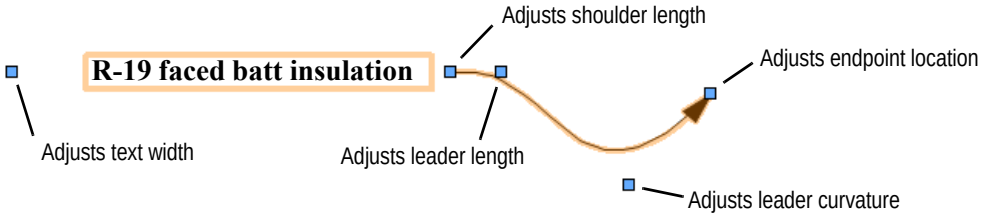
   To place callout text both above and below the line in the Split Circle **Bubble Style**, press Enter between the lines of text.

4. Click **OK**.
5. Set the text attributes as needed using the **Text > Format Text** command. You can also assign a text style to the callout.

Editing Callout Parameters

To edit the callout properties:

1. Select the callout object.
2. In the Object Info palette, change the parameters as desired. The parameters are described in “Inserting Callouts or Keynotes” on page 1293. An additional parameter that is available in the Object Info palette is **Leader Length**, which allows the length of the leader line to be specified precisely with numeric values rather than with the mouse. Finally, the marker type can be changed from the Attributes palette.
3. On the drawing, a callout object contains control points which can be moved with the mouse to change the callout text width, shoulder length, and endpoint position. A curved callout leader includes additional control points for adjusting the leader curvature.



Re-scaling Callout Objects

Callout objects are specified and drawn in page scale; they draw to the same apparent size, regardless of the layer scale. If the layer the callout object is on is re-scaled, or the callout object is cut and pasted between layers of different scales, the callout object automatically re-scales.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Callouts or Keynotes
The Attributes Palette
Formatting Text
Using Text Styles

Specifying Callout or Keynote Legend Bubble Shadow Settings

To specify bubble shadow edge and fill attributes:

1. In the Callout Preferences dialog box click **Shadow Settings**, or in the Object Info palette for a selected callout or keynote legend, click **Bubble Shadow Settings**.

   To activate this option, **Bubble Style** must be set to a style other than None, Bracket, ISO, or Vertical Accent Bar, and **Bubble Shadow** must be selected.

   The Bubble Shadow Settings dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Draw Bubble shadow edge/Draw Bubble shadow fill	Draws an edge/fill for the shadow
Use object’s pen color	Uses the callout object’s or keynote legend object’s pen color
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Use custom pen color	Uses a custom pen color; select the color

2. Specify the desired parameters and click **OK**.

Inserting Callouts or Keynotes

D Editing Keynotes and the Keynote Legend

Both the keynotes and keynote legend can be edited after placement. If only one keynote is to be edited, select it and make the changes in the Object Info palette. If the text of several keynotes is to be edited, select the keynote legend and make the edits from the Object Info palette.

Double-click a callout or keynote legend to edit it.

The keynotes bubble style can also display in the keynote legend; control the style with either the keynotes or the legend.

Editing Keynotes

To edit individual keynotes:

1. To change the text of a selected keynote, select the callout and click **Edit Note** in the Object Info palette.
2. Other callout parameters can be edited in the Object Info palette, like an ordinary callout object. For more information on the **Callout** tool, see “Inserting Callouts or Keynotes” on page 1293.
3. If there are several keynote objects, use the **Align/Distribute Leader Lines** command to improve readability (see “Aligning and Distributing Leader Lines” on page 1035).
4. After placement, a keynote can be converted into an ordinary callout object by deselecting **Place As Keynote** in the Object Info palette. The keynote is removed from the keynote legend.

Deleting Keynotes

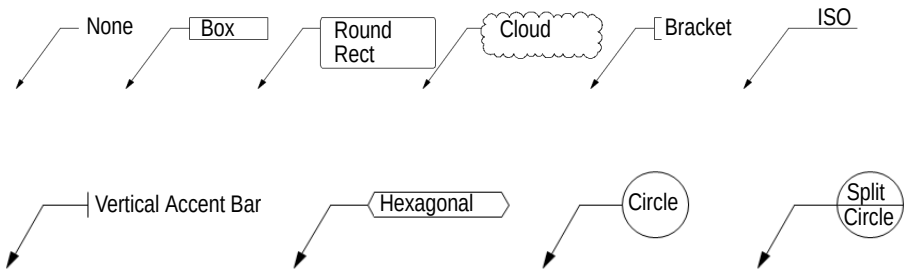
If you delete a keynote, the remaining keynotes are not renumbered. The keynote legend maintains a “gap” or placeholder displaying the custom text specified in **Unused Position Text**. However, if **Re-use Unused Positions** is selected and another keynote is placed in the drawing, the new keynote replaces the first “gap” keynote. Clicking **Remove Gaps** in the Object Info palette of a selected keynote legend also removes gaps; another way to remove gaps is to sort the keynotes in the Notes Manager: Keynote List dialog box.

Editing the Keynote Legend

To edit the keynote legend:

1. Edit the keynote legend parameters from the Object Info palette.

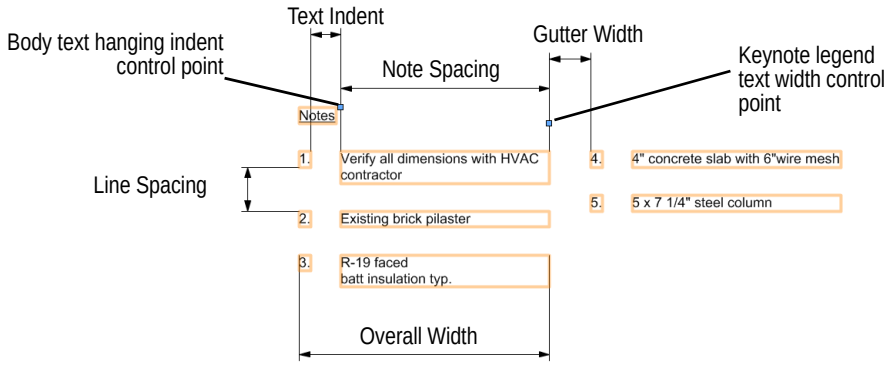
Click to show/hide the parameters.

Parameter	Description
Update	Refreshes the keynote legend display
Keynote Display	Select the display format for keynotes: numbers, letters, numbers or letters with the prefix and/or suffix that was entered in the callout preferences, or the description text from the notes database (click **Sort** in the Notes Manager:Keynote List to sort the notes as they will appear in the keynote legend) Because the note description can display in both the callouts and the keynote legend, it is possible to create an alphanumerical designation within the note description for each database entry, and display that number on the drawing. A database of notes with numeric descriptions can be used directly in the drawing, automatically sorted into the proper order.
Overall Width	Sets the text width for the entire keynote legend
Note Text Indent	Adjusts the hanging indent value for the legend body text
Note Spacing	Sets the width of the body of the keynote legend text
Line Spacing	Sets the number of lines as space between the notes
# of Columns	Select the number of columns for the keynote legend text
Gutter Width	When more than one column is selected for keynote legend text, sets the distance between the columns
Break Note Text Across Columns	When more than one column is selected for keynote legend text, allows text to break across columns; when deselected, text for each note does not break to the next column
Body Text Alignment	Specify how to align keynote legend body text. Horizontal text alignment for existing keynote legends can also be set by selecting **Text > Alignment** or **Text > Format Text** and selecting the desired alignment option.
Bubble Display	Sets the bubble display for keynote numbers/letters • None: No bubble displays for the keynote legend • per Callout: The bubble style used for the callout also displays in the legend • per Keynote Legend: The keynote legend controls the bubble style for callouts; select the **Bubble Style**
Bubble Style	When the **Bubble Display** is set to be controlled per Keynote Legend, specifies the bubble style to apply to the keynote legend and the keynotes 
RR Corner Radius	For Round Rect bubble styles, sets the corner radius
Text Margin	Sets the distance between the bubble and the text

Parameter	Description
Bubble Shadow	Select to draw the bubble with a drop shadow (does not apply to None, Bracket, ISO, or Vertical Accent Bar styles)
Bubble Shadow Settings	If **Bubble Shadow** is selected, opens the Bubble Shadow Settings dialog box, for specifying bubble shadow edge and fill attributes; see “Specifying Callout or Keynote Legend Bubble Shadow Settings” on page 1297
Re-use Unused Positions	When deselected, maintains a placeholder gap for deleted keynotes; new keynotes are appended to the end of the legend.When selected, replaces unused keynote positions, in order starting with the first one, as new keynotes are placed in the drawing.
Unused Position Text	Specifies the text to display in unused keynote positions
Remove Gaps	Removes unused keynote positions, renumbering the remaining positions
Edit Notes	Opens the Notes Manager:Keynote Legend dialog box, for editing the keynote text, description, or order A keynote object can be linked to a database by clicking **Show Database Controls** in the Notes Manager:Keynotes List dialog box.
Format Title Text	Opens the Format Text dialog box, for changing the text format of the legend title
Format Body Text	Opens the Format Text dialog box, for changing the text format of the legend body text

For information on formatting text, see “Formatting Text” on page 389. Text styles cannot be applied to keynote legends.

2. The keynote legend contains control points for adjusting the hanging indent and text width directly from the drawing.



3. To change keynote numbering or edit the text of all keynotes at the same time, select the keynote legend object and click **Edit Notes** in the Object Info palette.

   The Notes Manager:Keynote List dialog box opens.

   If the database controls are showing, click **Hide Database Controls**.

Click to show/hide the parameters.

Parameter	Description
Description	Lists the keynotes (by description) in the keynote legend in numbering order
Edit	Select a keynote description and click **Edit** to edit the note text or change the keynote description

Parameter	Description
Move Up/Move Down	Changes the order of the keynotes in the keynote legend; move the selected keynote up or down in the list
Sort	Sorts the notes in the legend alphanumerically; this is useful when the keynote legend **Keynote Display** is set to Note Description from Database, and the Description is set to an alphanumeric code (such as C101 or 032000). If there are any unused keynote positions, they are removed by this operation.
Note Text	Select a keynote description and edit the text of the note
<< Show Database Controls	Displays the database-related portion of the dialog box; when working without a database, these controls are not necessary (see “Placing Database Callouts” on page 1304) A keynote object can be linked to a database by clicking **Show Database Controls** in the Notes Manager:Keynotes List dialog box.

4. The keynotes are listed by a description. The description normally consists of the first 42 characters of the keynote, but this can be changed by selecting the description and clicking **Edit**.

   The Edit Note dialog box opens. To enter a different note description, deselect **Use the first 42 characters**, and change the **Description**. The note text can also be edited, if desired.

5. Click **OK** to close the Edit Note dialog box.
6. Click **OK** to close the Notes Manager:Keynote Legend dialog box and edit the keynote legend.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Callouts or Keynotes
Notes Management
Notes Management without Databases
Notes Management with Databases

Placing General Notes

Notes management includes a **General Notes** tool, for placing and numbering general notes.

To place a general notes object:

1. Click the **General Notes** tool from the Dims/Notes tool set.
2. Click the **Preferences** button on the Tool bar.

   The General Notes Tool Settings dialog box opens. Specify the default settings for placing General Notes during this session.

Click to show/hide the parameters.

Parameter	Description
Title	Specifies the name for the general notes block, or use the default General Notes title
Indent/Width/Spacing	Indicates the default formatting for the notes
General Notes Display	Select the display format for general notes: numbers, letters, or the description text from the general notes database
Body Text Alignment	Specify how to align general notes body text
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Show Databases	Deselect to place general notes without a database
Select Title Text/Body Text Format	Specifies the default text formatting for the title and note text

3. Click to place the general notes object in the drawing.

   The Notes Manager:General Notes dialog box opens.

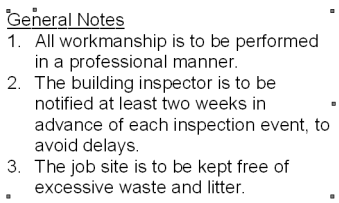
Click to show/hide the parameters.

Parameter	Description
Description	Lists the general notes (by description) in numbering order
Add	Opens the Add Note dialog box, for adding a general note and specifying its description
Remove	Deletes the selected general note(s)
Remove All	Deletes all general notes
Edit	Select a general note and click **Edit** to edit the note text or change the general note description
Move Up/Move Down	Changes the order of the general notes; move the selected general note up or down in the list
Note Text	Select a general note description and edit the text of the note
<< Show Database Controls	Displays the database-related portion of the dialog box; when working without a database, these controls are not necessary A general note object can be linked to a database by clicking **Show Database Controls** in the Notes Manager:General Notes dialog box. See “Notes Management with Databases” on page 1304.

4. To add a new general note, click **Add**.

   The Add Note dialog box opens.

5. Enter the new general note text. The general notes are listed by a description. The description normally consists of the first 42 characters of the general note. To enter a different note description, deselect **Use the first 42 characters**, and change the **Description**.
6. Click **OK** to add the new general note.
7. The note text can be edited, if desired. Select the note description and edit the text in the Note Text area. The note text and its description can be edited by clicking **Edit**. Delete selected notes by clicking **Remove**.
8. Order the general notes by clicking **Move Up** or **Move Down**.
9. Click **OK** to place the general notes object.



Editing General Notes
Notes Management

Editing General Notes

To edit a general notes object:

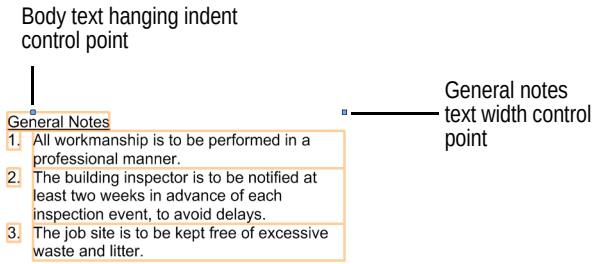
1. The general note parameters can be edited from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Title	Specifies the general notes title
Indent	Adjusts the hanging indent value for the general notes body text
Width	Sets the width of the general notes text
Spacing	Sets the spacing between each note
General Notes Display	Select the display format for general notes: numbers, letters, or the description text from the general notes database
Body Text Alignment	Specify how to align general notes body text. Horizontal text alignment for existing general notes can also be set by selecting **Text > Alignment** or **Text > Format Text** and selecting the desired alignment option.
Edit Notes	Opens the Notes Manager:General Notes dialog box, for editing the general notes text, description or order A general note object can be linked to a database by clicking **Show Database Controls** in the Notes Manager:General Notes dialog box. See “Notes Management with Databases” on page 1304.
Format Title Text	Opens the Format Text dialog box, for changing the text format of the general notes title
Format Body Text	Opens the Format Text dialog box, for changing the text format of the general notes body text

For information on formatting text, see “Formatting Text” on page 389 and “Using Text Styles” on page 393.

2. The general notes object contains control points for adjusting the hanging indent and text width directly from the drawing.



Placing General Notes
Notes Management
Aligning and Distributing Leader Lines

Notes Management with Databases

Instead of manually typing the text for every callout or keynote and general notes object, text that is used repeatedly—by one or many users—across projects and files can be stored in a tab-delimited text file (referred to as a "database"). The use of a database for storing notes saves time and prevents errors, and standardizes compliant notes usage across files.

There is no limit to the number of databases that can be used for annotating a drawing. The databases can have an unlimited number of notes of any length. After placement, notes can be easily updated if changes are made to a database.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Placing Database Callouts

Callouts with database notes are placed with the extended capabilities of the **Callout** tool.

To place a database callout:

1. Click the **Callout** tool from the Basic palette.
2. Click the **Preferences** button on the Tool bar.

   The Callout Preferences dialog box opens. For more information on the **Callout** tool, see "Inserting Callouts or Keynotes" on page 1293.
3. Select **Get Text from Database**; the current database file is displayed as a button.
4. To select a different database, click the database button.

   The standard Open dialog box allows a different database file to be selected. Select the database file, and click **Open**.

   A sample database is located in [Vectorworks]\Libraries\Defaults\Notes\Callouts.txt
5. Click **OK**.
6. Click in the drawing to place the callout object.

   Once the callout leader is drawn, the Notes Manager:Callout dialog box opens automatically, displaying the information from the selected database.

   The notes in a selected section are listed by a description. The description normally consists of the first 42 characters of the note.

Click to show/hide the parameters.

Parameter	Description
Active Database	Displays the currently selected database file
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
New	Creates a new, blank database
Choose	Selects a different, existing database. A database from a previous version of the Vectorworks program can be specified, but it must be converted to the current version format as prompted.
Save As	Saves the current database with a new name
Sections in Database	Lists the note sections; the associated notes in a selected section are displayed in **Notes in Database**
Add	Adds a new note section; enter the section name in the Enter String dialog box
Remove	Deletes the currently selected section and its associated notes
Edit	Renames the currently selected section; enter the new name in the Enter String dialog box
Notes in Database	Lists the notes associated with the current database section
Add	Opens the Add Note dialog box, for adding a note to the database section
Remove	Deletes the currently selected note
Edit	Opens the Edit Note dialog box, for editing the selected note or its description
Note Text	Displays the note text of the currently selected note; the note text can be edited
Database Filter	Filters the database sections, descriptions, or notes with a search term, to more easily locate a note
Edit	Opens the Edit Filter List dialog box, to add or edit database filters
Prefs	Sets the automatic save options for the database
Update >	Moves the selected note to the Callout Text list for placement on the drawing
< Update	Replaces the database with the text in the Callout Text list
< Add	Adds the text in the Callout Text list to the database, placing it in the current database section
< Locate	If the selected text in the Callout Text list originated from a database, locates the note's database, section, and description
Connect to Database with:	Sets how to connect to the selected database; if an absolute path is selected, the database is expected to be present at the specific location. If a relative path is selected, the database is expected to be present at the indicated location relative to the current file.
Hide Database Controls >>	Hides the database portion of the callout dialog box; edits made with the database hidden are not reflected in the database

7. To add a note to a selected section, click **Add** under **Notes in Database**.

   The Add Note dialog box opens. Specify the section, description, and note text.

Click to show/hide the parameters.

Parameter	Description
Section	Select the associated section for the note
Description	By default, consists of the first 42 characters of the note text; deselect **Use the first 42 characters**, and enter a different note description, if desired

Parameter	Description
Note Text	Type the new note text
Use the first 42 characters	If the default description is not desired, deselect and enter a new **Description**

8. Click **OK** to add the note to the specified section and return to the Notes Manager:Callout dialog box.
9. Change the note description by selecting the description and clicking **Edit**.

   The Edit Note dialog box opens. To enter a different note description, deselect **Use the first 42 characters**, and change the **Description**. The note text can also be edited, if desired, and the note can be associated with a different section.
10. Click **OK** to edit the note description and return to the Notes Manager:Callout dialog box.
11. The text of a selected note can be edited in **Note Text**.
12. To search easily for a note, section, or description, use a filter. Only notes, sections, or descriptions with the specified text are displayed. Select the desired filter from the **Database Filter** list in the Notes Manager:Callout dialog box. To display all notes, sections, and descriptions, select **All Notes**. In addition, if no items match a selected filter, all notes are displayed.
13. To edit the database search filters, click **Edit** under **Database Filter**.

    The Edit Filter List dialog box opens. The filters are listed in order.

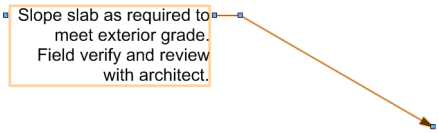
Click to show/hide the parameters.

Parameter	Description
Filter list	Lists the search filters in the order they display in the database filter list
Add	Adds a search filter to the list; enter the new term in the Enter String dialog box
Remove	Deletes the currently selected search filter(s)
Edit	Renames the current search filter; enter the new name in the Enter String dialog box
Move Up/Move Down	Changes the filter list order; select the search filter and move it up or down in the list

14. Click **OK** to change the search filters and return to the Notes Manager:Callout dialog box.
15. Changes and additions to the notes database are saved automatically according to the preferences specified. Click **Prefs** to set these preferences.

    The Database Preferences dialog box opens. Set the desired actions that automatically save database changes by selecting the associated check box(es). If no automatic saving is specified, you will be prompted frequently about saving the database.

    The **Locate** and **Hide Database** options are available when editing a General Notes or Keynote Legend with database.
16. Click **OK** to return to the Notes Manager:Callout dialog box.
17. Once the desired note has been entered or located, select it and click **Update** to move it to the Callout Text list.
18. Click **OK**. The selected note is used for the callout text.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing a Database Callout
Notes Management with Databases
Notes Management

Editing a Database Callout

The database callout can be edited after placement.

To edit a database callout:

1. To change the note text, select the callout and click **Edit Note** in the Object Info palette.
2. The Notes Manager:Callout dialog box opens. Select a different note as described in “Placing Database Callouts” on page 1304, and click **OK**.
3. Other callout parameters can be edited in the Object Info palette as for an ordinary callout object. For more information on the **Callout** tool, see “Inserting Callouts or Keynotes” on page 1293.
4. If there are several callout objects, use the **Align/Distribute Leader Lines** command to improve readability (see “Aligning and Distributing Leader Lines” on page 1035).
5. After placement, a database callout can be converted into an ordinary callout object by clicking **Edit Note** in the Object Info palette, and then clicking **Hide Database Controls**. The callout text can then be edited, and the database text is not affected.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Placing Database Callouts
Notes Management with Databases
Notes Management

Placing Database Keynote Legends

Keynote legends can take advantage of a notes database.

To place a database keynote legend:

1. Click the **Callout** tool from the Basic palette.
2. Click the **Preferences** button on the Tool bar.

   The Callout Preferences dialog box opens. As described in “Inserting Callouts or Keynotes” on page 1293, select **Place As Keynote** and click **OK**.
3. Place the callouts in the drawing area.
4. Select the keynote legend. In the Object Info palette, click **Edit Notes**.
5. The Notes Manager:Keynote List dialog box opens.
6. If necessary, click **Show Database Controls** to access the keynote legend database parameters.

   The dialog box is divided into two sections, with the database information on the left and the keynote legend information on the right. The buttons in the center of the dialog box move information from the database to the legend and vice-versa.

Click to show/hide the parameters.

Parameter	Description
Active Database, Sections in Database, Notes in Database, Note Text, Database Filter, and Prefs	Edits the database sections, notes, note text, search filters, and saving preferences as described in “Placing Database Callouts” on page 1304
Notes in the Selected Keynote List Object	Edits the keynote legend notes and description as described in “Inserting Callouts or Keynotes” on page 1293; the **Relative Path** column indicates whether the associated database is referenced by a relative path (check mark) or absolute path (no check mark)
Update >	Overwrites the selected keynote legend note(s) with the selected database note
< Update	Overwrites the selected database note with the selected keynote legend note
< Add	Adds the selected keynote legend note(s) to the database, placing the note(s) in the current database section
Auto-Locate	Automatically locates the database, section, and description of a selected keynote legend note, if the note originated from a database
< Locate	If the selected keynote legend note originated from a database, locates the note’s database, section, and description
Hide Database Controls >>	Hides the database portion of the keynote legend dialog box; edits made with the database hidden are not reflected in the database

7. Click **OK** to change the keynote legend, replacing the notes with database notes.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Notes Management with Databases
Editing Keynotes and the Keynote Legend
Notes Management

Placing Database General Notes

General notes can take advantage of a notes database.

To place a database general notes object:

1. Click the **General Notes** tool from the Dims/Notes tool set.
2. Click to place the general notes object in the drawing.

   The Notes Manager:General Notes dialog box opens.
3. Click **Show Database Controls** to access the general notes database parameters.

   The dialog box is divided into two sections, with the database information on the left and the general notes information on the right. The buttons in the center of the dialog box move information from the database to the general notes and vice-versa.

   A sample database is located in [Vectorworks]\Libraries\Defaults\Notes\GeneralNotes.txt

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Active Database, Sections in Database, Notes in Database, Note Text, Database Filter, and Prefs	Edits the database sections, notes, note text, search filters, and saving preferences as described in “Placing Database Callouts” on page 1304
Notes in the Selected General Notes Object	Edits the general notes and description as described in “Placing General Notes” on page 1301; the **Relative Path** column indicates whether the associated database is referenced by a relative path (check mark) or absolute path (no check mark)
Add >	Adds one or more selected database notes to the general notes
Update >	Overwrites the selected general note(s) with the selected database note
< Update	Overwrites the selected database note with the selected general note
< Add	Adds the selected general note(s) to the database, placing the note(s) in the current database section
Auto-Locate	Automatically locates the database, section, and description of a selected general note, if the note originated from a database
< Locate	If the selected general note originated from a database, locates the note’s database, section, and description
Hide Database Controls >>	Hides the database portion of the general notes dialog box; edits made with the database hidden are not reflected in the database

4. The note in the **Notes in the Selected General Notes Object** list will be placed as a general note. Click **OK** to place the general note.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Text to a Database
Notes Management with Databases
Placing General Notes
Editing General Notes
Notes Management

Adding Text to a Database

Text objects can be added from the drawing to a selected database. This is convenient for retrieving text from older, converted files for use with the Notes Manager or when adding frequently-used text to a database for easy future insertion as an annotation.

To add drawing text to a database:

1. Select **Text > Add Text to Database**.

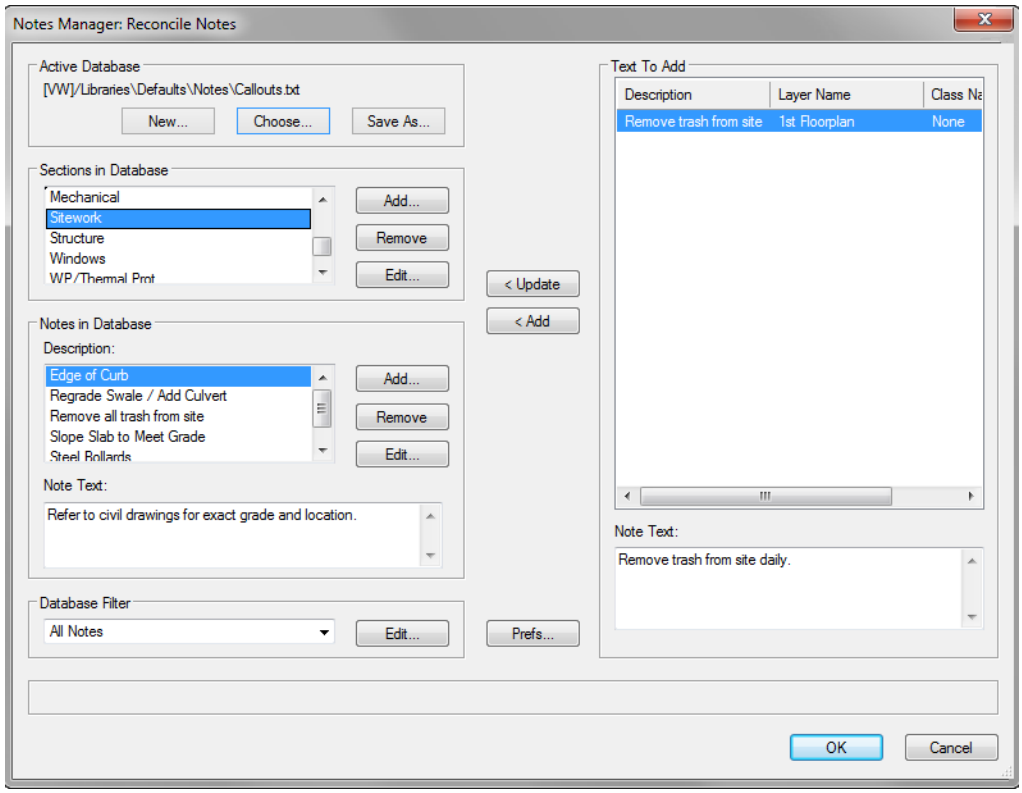
   The Add Text to Database dialog box opens. Specify which text objects to select and indicate in which databases to search for duplicated text.

Click to show/hide the parameters.

Parameter	Description
Add Text Meeting These Criteria	Specifies the criteria for selecting text objects to be added to a database
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Selected Text Only	Adds only selected text to the database; when deselected, adds text from all text objects in the file
Layers/Classes	Adds only text from the specified layers and/or classes to the database; when deselected, adds text from text objects in all layers or classes in the file
Exclude Text Already in One of These Databases	Lists the databases in which to search for duplicated text; the text search is case sensitive and the match must be exact. If duplicated text is found, it is not added to the database.
Add	Adds a database to the text search database list
Change	Allows changes to a selected database's location
Remove	Removes a database from the text search database list

2. Click **OK**. If any of the text was not found in the text search database(s), the Notes Manager:Add Text to Database dialog box opens.



3. The text to be added is listed on the right. As described in "Placing Database Keynote Legends" on page 1307, the text can be added to the active database in the desired section. Click **OK**.

Placing Database General Notes
Reconciling Database and Drawing Notes
Notes Management with Databases
Notes Management

Reconciling Database and Drawing Notes

The **Reconcile Notes** command handles differences between the database and the notes in the drawing. For example, if database callouts have been placed and the database is edited afterwards, differences must be reconciled.

To reconcile notes:

1. Select **Text > Reconcile Notes**.

   The Reconcile Notes dialog box opens. Select the note objects to reconcile.

Click to show/hide the parameters.

Parameter	Description
Callouts obtained from database	Selects database callouts
Callouts not obtained from database	Selects ordinary callouts and keynote callouts that are not linked to a database
Callouts that used to get their text from a database	Selects callouts that were once database callouts and obtained note text from a database
Callouts that never got their text from a database	Selects callouts that did not obtain note text from a database
General Notes objects	Selects general notes
Layers / Classes	Select the layers and classes with notes to include for reconciliation

2. Click **OK** to reconcile the notes. If any of the notes reference databases, the Notes Manager attempts to locate those databases. It searches first in the same folder as the current file; if not found, it searches in the same place relative to the Vectorworks program executable. If the databases still cannot be found, the Notes Manager tries to locate them with fully-qualified paths.

   Place database files from another user in the same folder as the drawing file so that the Notes Manager can easily locate the databases.

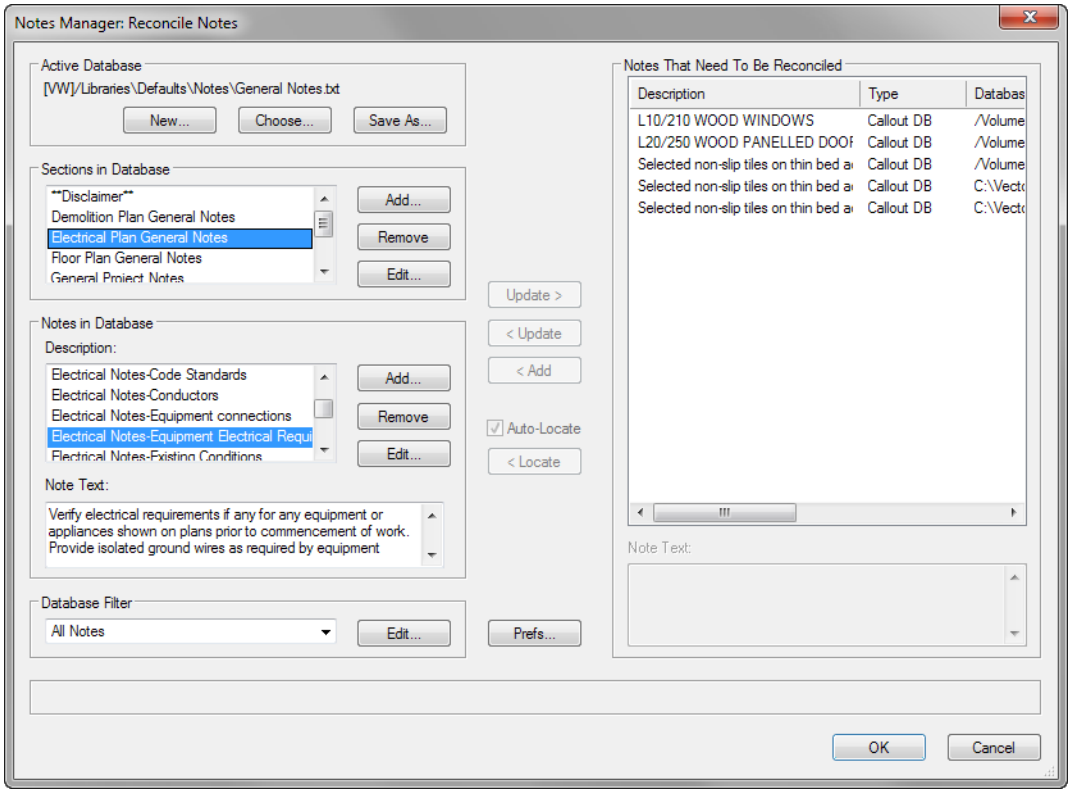
3. If a database referenced by notes cannot be located, the Cannot Locate Database(s) dialog box opens. For each database that cannot be located, click **Browse** to specify the database location.
4. Click **OK**. Next, the Notes Manager searches the database(s) for the notes.
5. If one or more notes cannot be located in the database, an alert dialog box opens asking if you want Notes Manager to search for these notes in a database search list.

   If there are no additional database files to be specified, click **No** and proceed to step 8. If the notes could be located in another database, click **Yes** to specify the database location.

6. The Specify Database Search List dialog box opens. Click **Add** to specify additional databases where the notes are located. Click **Remove** to delete a selected database from the list. The number of notes which have been located is displayed.
7. Click **OK**.
8. If some notes have still not been located, the Notes Manager:Reconcile Notes dialog box opens.

   The dialog box is divided into two sections, with the database information on the left and the notes to be reconciled on the right. The buttons in the center of the dialog box move information from the database to the general notes and vice-versa.

   By using the center buttons, reconcile each note manually. As a note is reconciled, it is removed from the list.



Click to show/hide the parameters.

Parameter	Description
Active Database, Sections in Database, Notes in Database, Note Text, Database Filter, and Prefs	Edits the database sections, notes, note text, search filters, and saving preferences as described in “Placing Database Callouts” on page 1304
Notes That Need To Be Reconciled	Displays the note text, type of note, database name and location, layer and class name, and database reference type (relative or absolute) for the notes that have not been reconciled
Update >	Overwrites the selected note(s) to be reconciled with the selected database note
< Update	Overwrites the selected database note with the selected note to be reconciled
< Add	Adds the selected note(s) to be reconciled to the database, placing the note(s) in the current database section
Auto-Locate	Automatically tries to locate the database, section, and description of a selected note to be reconciled, if the note originated from a database
< Locate	If the selected note to be reconciled originated from a database, tries to locate the note’s database, section, and description

9. Click **OK**.

   An alert message explains how the notes were reconciled.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Notes Management with Databases
Notes Management
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

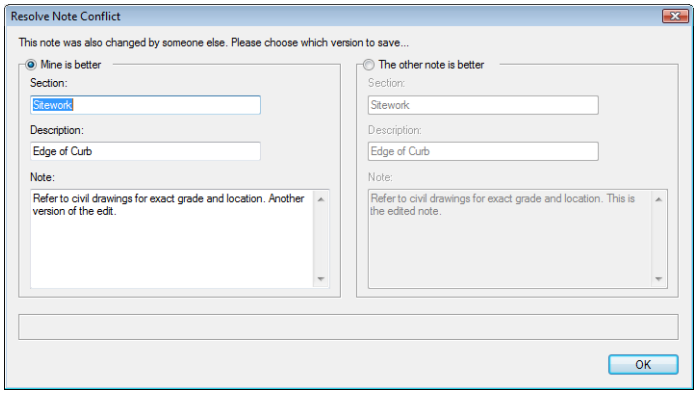
Locating Databases on a Network

To use databases efficiently in a multiple-user environment, the database files should be placed on a network. There, they can be accessed concurrently by several users who will then conform to office standards and will not need to retype notes.

> Windows users should choose databases by browsing from Network Neighborhood (My Network Places) rather than from mapped drive letters. If databases are selected from mapped drives, the database references are then dependent on drive mapping.

Database changes made by different users are merged automatically.

A conflict could arise if two different users edit the same note in the same database at the same time. If this occurs, the Resolve Note Conflict dialog box opens when closing the Notes Manager dialog box.



On the left, your version of the note displays, and on the right, the other user's version displays. Select which version of the note to save by selecting **Mine is better** or **The other note is better**. Click **OK** to resolve the conflict.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Notes Management with Databases
Notes Management

Converting Notes from Previous Versions

Databases and notes created in Vectorworks software versions prior to version 11 can be converted for use in the current version.

Databases created in previous versions of the Notes Manager are automatically converted to the current version when they are selected by clicking **Choose** to select a database.

Callouts created in version 9 and above are automatically converted to the current format. They can be used as ordinary callouts, keynotes, or database callouts.

Other notes objects, such as Text Notes, Keynotes, and General Notes created by previous Notes Manager versions, as well as Vectorworks text objects, must be converted by command. Vectorworks text objects are converted to general notes.

To convert old objects to the current format:

1. Select **Text > Convert Old Notes to New**.
2. The Convert Old Notes to New dialog box opens. Select the type of old notes to convert.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
All updatable objects	Converts any note-related items in the drawing
All selected objects	Converts any note-related items in the current selection
Just the following objects	Select the type of objects to be converted (Text Notes, Keynotes, General Notes from version 8.52 to 10, and text objects)
Layers/Classes	Specifies the layers and classes to be searched for objects to convert

3. Click **OK** to convert the objects to the current version Notes Manager format.
4. The Conversion Results dialog box displays a summary of the objects converted.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Notes Management with Databases
Notes Management
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Worksheets

The worksheet functionality in the Vectorworks program complements its drawing functionality, making it a complete package for your work process. From the information present in the file, you can create worksheets to track data, create cost and material lists, perform calculations, and more. Worksheets are integrated within the Vectorworks file, which eliminates the need for a separate program and reduces the number of files per project.

Worksheets include both database and spreadsheet functionality. Data can be obtained from the drawing, and then calculations can be performed on that data.

Worksheets can be imported and exported, which allows data to be shared between worksheets, files, and other spreadsheet programs. A worksheet can also be added to a drawing and printed.

To get a quick overview of worksheet features, see “Worksheet Tutorial: Creating a Wall Schedule” on page 1361.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Worksheets

For complex drawings, it is best to create separate worksheets for each task rather than one large worksheet. Worksheets can be linked to share data, formulas, and calculations.

Worksheets can be created in several ways:

- Use the **Create Report** command to select worksheet data from the information associated with the objects in the drawing. See “Creating Reports” on page 1316.
- Use the Resource Browser to create a blank worksheet, and then add the desired information to it. See “Creating a Blank Worksheet” on page 1317.
- Import worksheets from other Vectorworks files or from other spreadsheet programs. See “Importing Worksheets” on page 1358.
- If Vectorworks Design Series is installed, preformatted records and schedules such as room finishes, plant lists, and lighting instruments can be added to the drawing. See “Creating Schedules” on page 1864 and “Creating Schedules Automatically” on page 1318.

Once created, a worksheet is saved with the file and is listed in the Resource Browser. It can also be accessed by selecting **Window > Worksheets**.

If the same objects are typically used in your drawings, you can create a template file with a worksheet that serves as a “master price list” listing all the objects and their costs. Then, to create materials lists and cost estimates for a new design, simply import the worksheet into the new drawing file.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Reports

The **Create Report** command allows you to select data that is attached to objects in a drawing (such as manufacturer, size, and price) and create a worksheet from it. The command can either create a new worksheet or append database rows to an existing worksheet. For more information about how to attach data to objects, see “Record Formats” on page 262.

To create a report from objects in a drawing:

1. Select the **Create Report** command from the appropriate menu:
   - Fundamentals, Architect, Landmark workspaces: **Tools > Reports > Create Report**
   - Spotlight workspace: **Spotlight > Reports > Create Report**

   The Create Report dialog box opens. Specify the report criteria. Items in the **Worksheet Columns** list are listed in the order in which they will appear in the worksheet; to change the order, click in the # column and drag the item to the desired position in the list.

Click to show/hide the parameters.

Parameter	Description
Title	Enter a worksheet title; if the report data is appended to an existing worksheet, this title is added to a spreadsheet cell above the database rows that are appended
List all	Select whether to list all symbols in the drawing, or to list all drawing objects that have a specific record attached to them
Listing objects with record	If you selected the option to list all objects with a specific record, select the record format to use
Possible Columns	Lists the columns that can be selected for inclusion in the worksheet
Worksheet Columns	Lists the columns that are currently selected for inclusion in the worksheet
Add >> / Add All >>	To add a column to the worksheet, select it in the **Possible Columns** list and click **Add**; click **Add All** to move all the columns to the **Worksheet Columns** list
<< Remove / << Remove All	To remove an item from the worksheet, select it in the **Worksheet Columns** list and click **Remove**; click **Remove All** to move all the columns to the **Possible Columns** list
Summarize items with the same	Select the checkbox to summarize symbols or objects that contain the same value for a specified field, rather than creating a new database sub-row for each; then select the field to summarize from the drop-down list
Options	Opens the Create Report Options dialog box

2. Click **Options** to specify additional report criteria.

   The Create Report Options dialog box opens.

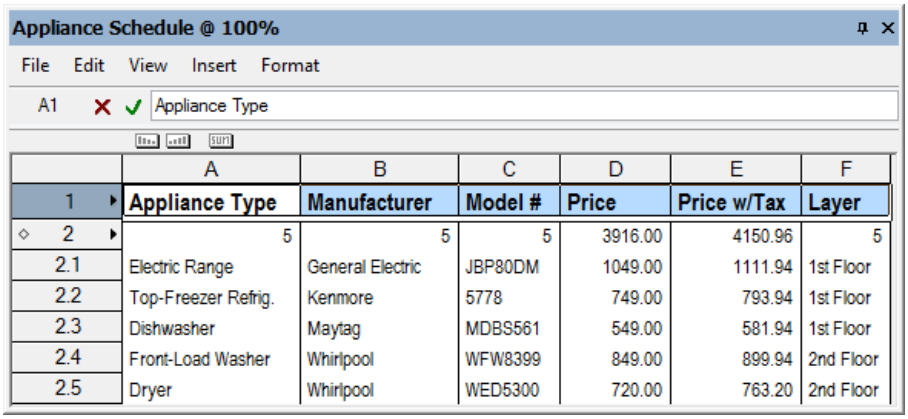
Click to show/hide the parameters.

Parameter	Description
New worksheet	Select to create a new worksheet
Append to existing worksheet	Select to append the information to a worksheet, and then select the target worksheet
Search in symbols	Select to search inside symbol instances for embedded objects or symbols and include this information in the worksheet

Parameter	Description
Search in plug-in objects	Select to search inside plug-in object instances for embedded objects or symbols and include this information in the worksheet

3. Specify the options for creating the report, and click **OK** to return to the Create Report dialog box.
4. Click **OK** to create the worksheet.

   The worksheet opens automatically. The top row of the worksheet contains a title for each column selected. Next is a database header row (indicated by a diamond next to the row number) that contains sub-row totals for each column. Beneath the header row are sub-rows for each object or symbol in the drawing that matches the report criteria.



	A	B	C	D	E	F
1	Appliance Type	Manufacturer	Model #	Price	Price w/Tax	Layer
◇ 2	5	5	5	3916.00	4150.96	5
2.1	Electric Range	General Electric	JBP80DM	1049.00	1111.94	1st Floor
2.2	Top-Freezer Refrig.	Kenmore	5778	749.00	793.94	1st Floor
2.3	Dishwasher	Maytag	MDBS561	549.00	581.94	1st Floor
2.4	Front-Load Washer	Whirlpool	WFW8399	849.00	899.94	2nd Floor
2.5	Dryer	Whirlpool	WED5300	720.00	763.20	2nd Floor

5. To add more data to the worksheet, repeat steps 1 through 4 and select the **Append to existing worksheet** option.
6. Once all the data is added, edit the worksheet as needed. For example, add rows or columns, change the text format, or add color. To hide the database header rows, toggle the **Database Headers** setting on the **Worksheet** menu.

   Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Blank Worksheet
Using Worksheets
Importing Worksheets

Creating a Blank Worksheet

Instead of using the **Create Report** command, you can create a blank worksheet and add data to it manually. This gives you more control over the contents and format of the worksheet.

To create a blank worksheet:

1. Select **Window > Palettes > Resource Browser**.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Worksheet**.

   The Create Worksheet dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Name	Enter a worksheet name. To rename the worksheet later on, select the worksheet in the Resource Browser, and click **Resources > Rename**.
Rows / Columns	Enter the number of rows and columns for the worksheet; the number of rows and columns can be modified later

4. Specify the basic worksheet parameters and click **OK**.

   A new worksheet window opens.

5. At this point, all rows contain spreadsheet cells, and they are all undefined. Define the contents of each row and cell as needed:

   - To add simple text, numbers, or formulas to the worksheet, see “Entering Data in Spreadsheet Cells” on page 1333.
   - To list data that is associated with objects in the drawing, change a spreadsheet row into a database row, and specify which objects to include in the list. A sub-row displays for each object that matches the criteria defined in the database header row. Then specify which information to display in the columns for each row; these can be fields from the object’s data record, as well as text, numbers, or formulas. See “Entering Data in Database Rows” on page 1339.
   - To add images to either spreadsheet or database rows, see “Inserting Images in Worksheets” on page 1343 (Vectorworks Design Series required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Reports
Using Worksheets

Creating Schedules Automatically

The Vectorworks Landmark and Spotlight products include a **Choose Schedule** command with preformatted schedules that are generated automatically. The Vectorworks Architect product includes preformatted, customizable records and schedules as described in “Records and Schedules” on page 1859. The schedules can be created any time; as objects are added to the drawing, recalculate the worksheet to update the results.

The preformatted schedules are from the default content included with the Vectorworks Design Series products in the [Vectorworks]\ Libraries\Defaults\Reports~Schedules folder (see “Resource Libraries” on page 219).

The default schedules for Vectorworks Landmark include plant lists, irrigation plans, and existing tree reports, with or without images. Plants to be included in the Plant List worksheets must have **On Plant List** selected in the Plant Settings dialog box or the Object Info palette.

A formatted worksheet can be saved as a custom schedule to be used again; export the worksheet resource (renamed with a custom name) to a file in the Reports~Schedules folder in your user folder. Your customized worksheet becomes available for selection in the Choose Schedule dialog box. For information about exporting resources, see “Exporting Custom Resources” on page 234.

> Another way to create a schedule is to select the **Create Report** command. List the objects with the relevant record, such as the Existing Tree record, and then select the desired columns to include in the report. See “Creating Reports” on page 1316 for more information.

To add a schedule:

1. Select a command from the appropriate menu:

   - Landmark workspace: **Tools > Reports > Choose Schedule**
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Spotlight workspace: **Spotlight > Reports > Choose Schedule**
- Architect workspace: **Tools > Reports > VA Create Schedule**

The Choose Schedule dialog box (for Landmark and Spotlight) or Create Schedule dialog box (for Architect) opens. Select one of the worksheets to create; select **Place worksheet on drawing** to add the worksheet to the drawing area.

2. Click **OK** to create the selected worksheet.

   If the selected schedule already exists in the file, a warning dialog box opens. Select whether to replace or rename the new schedule (some schedules also have a recalculate option), and click **OK**.

3. If the worksheet is to be placed on the drawing, click to indicate the position of the top left corner of the worksheet.

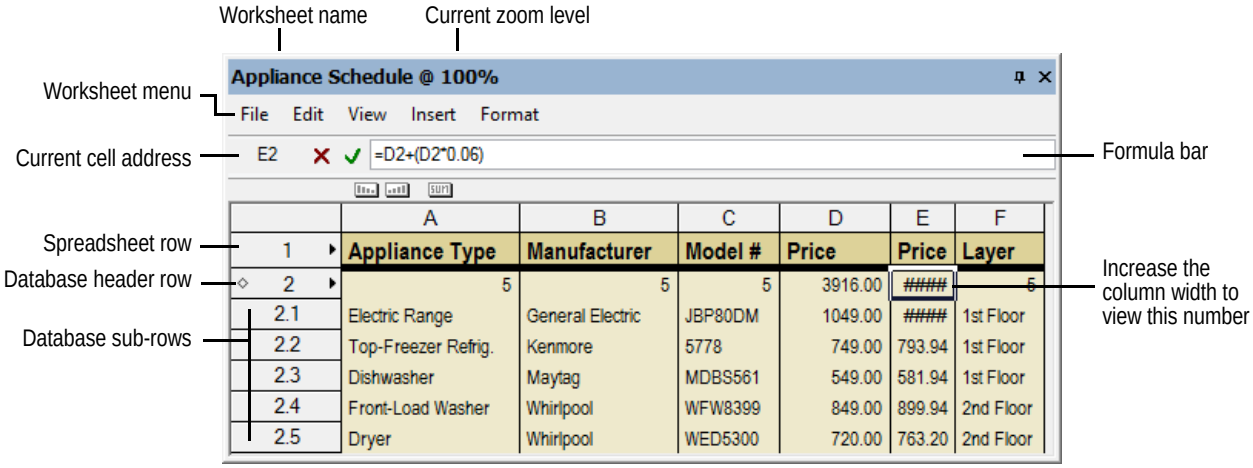
   The worksheet, populated with specific information from the current drawing, is automatically created.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Worksheets
Resource Libraries
Adding Plants to the Design
Placing Existing Trees
Irrigation
Records and Schedules

Using Worksheets

Worksheets can obtain data from the drawing based on specified criteria, and then list the data and allow calculations to be performed on the data.

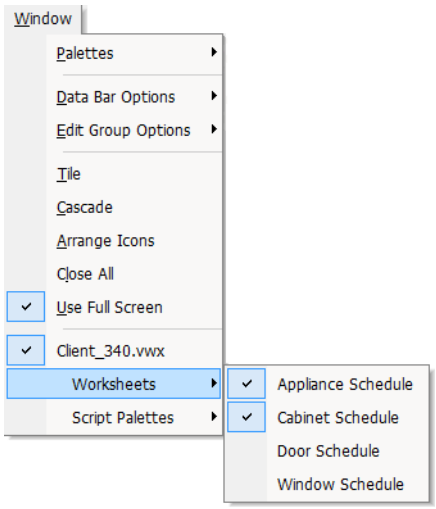


Worksheets can have two types of rows: spreadsheet and database. The cells in a spreadsheet row contain constants (text or numbers), or formulas. Database rows consist of a header row and sub-rows, and they show data that are associated with specific drawing objects. The database header row is marked with a diamond shape next to the row number. Set selection criteria for this row, and a sub-row is created for each object that meets the criteria. In this example, the database header row 2 is set to list each object in the drawing that has an “appliance record” attached to it. The sub-rows 2.1 through 2.5 represent the five objects in the drawing that meet this criteria.

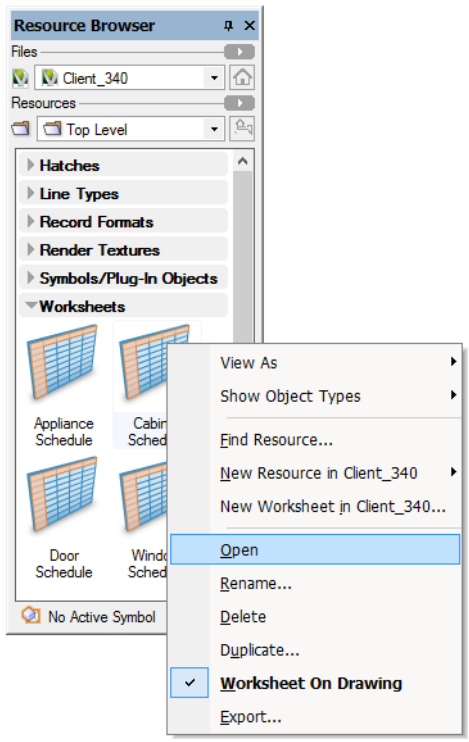
Rows are numbered sequentially starting with 1, and columns are labeled alphabetically starting with A. Database sub-rows are numbered with the database header row’s number, followed by a decimal and sequential numbers (header row 2 has sub-rows 2.1, 2.2, 2.3, and so on). The cell’s column letter and row number indicate the spreadsheet cell address, as in A4 or D2 (database sub-rows display a blank address).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

When worksheets exist in an open file, the **Window > Worksheets** command becomes available. All the worksheets present in the indicated file are listed. Worksheets with a check mark are currently open. To open a worksheet, select it from this menu, or select the worksheet from the Resource Browser and then select **Resources > Open**.

Selecting a worksheet from the Window menu



Selecting a worksheet from the Resources menu



A worksheet opens in a separate window, which can be resized, moved, and closed. The worksheet window contains its own menu and context menus.

Because an open worksheet is in a separate window, it is not printed with the drawing. To include a worksheet as part of a drawing, select the worksheet in the Resource Browser and click **Resources > Worksheet on Drawing**. When the worksheet is open, the worksheet on the drawing displays as an "X." When the worksheet is closed, the updated worksheet displays on the drawing. Double-click the worksheet from the drawing to open it. See "Worksheets as Graphic Objects" on page 1360.

Use the **Format Cells** command to format individual rows, columns, and even cells of the worksheet as needed. The format is retained when the worksheet is included on the drawing. Alternatively, with the worksheet window closed, select the worksheet object on the drawing, and use the Attributes palette to modify the fill, pen, and line thickness attributes for the entire object.

Appliance Type	Manufacturer	Model #	Price	Price w/Tax	Layer
Electric Range	General Electric	JBP80DM	1049.00	1111.94	1st Floor
Top-Freezer Refrig.	Kenmore	5778	1600.00	1696.00	1st Floor
Dishwasher	Maytag	MDBS561	549.00	581.94	1st Floor
Front-Load Washer	Whirlpool	WFW8399	849.00	899.94	2nd Floor
Dryer	Whirlpool	WED5300	510.00	540.60	2nd Floor

Selecting and Editing Worksheet Cells
Worksheet Commands
Formatting Worksheet Cells

Selecting and Editing Worksheet Cells

The contents of cells can be edited, and the rows and columns can be resized, moved, cut, copied, and pasted.

Moving Around in Worksheets

The following table describes the keys used to move around in the worksheet.

Keys	Description
Arrow (Up, Down, Right, Left)	Moves by one cell in the direction indicated
Tab	Moves right by one cell
Enter	Moves down by one cell
Shift+Tab	Moves left by one cell
Shift+Enter	Moves up by one cell

If more than one cell is selected, movement is restricted to the selected cells only.

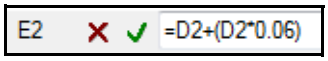
Selecting Cells

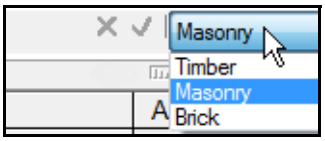
Selected cells are surrounded with a black outline. When multiple cells are selected, the active cell is white, and the remaining selected cells are highlighted in blue.

Selection	Action
A single cell	Click on the cell
A range of cells	Click-drag across a range of cells to select them, or click in one corner and Shift-click in the opposite corner
An entire column or row	Click the column letter or row number; to select multiple rows or columns, click-drag across the column letters or row numbers, or click the first column letter or row number, and then Shift-click the last column letter or row number in the desired range
Non-contiguous cells, rows, or columns	Press and hold the Ctrl (Windows) or Command (Mac) key and then click on each cell, row, or column to select
The entire worksheet	Click the empty box directly above the row number boxes

Editing Cell Contents

When a cell is selected, the display of the Formula bar indicates whether the contents of the cell can be edited.

Formula Bar Display	Explanation
 The cell address displays, and the red X and green check mark are active	The cell is in a spreadsheet row or database header row. Type directly in the Formula bar to enter text, numbers, or a formula. To accept the edits and change the cell contents, click the green check mark. To cancel the edits, click the red X.
 No cell address displays; the current cell value is not editable	In database sub-rows, the results of calculations cannot be edited. In addition, object attribute information, such as the class the object belongs to, cannot be edited in the worksheet.

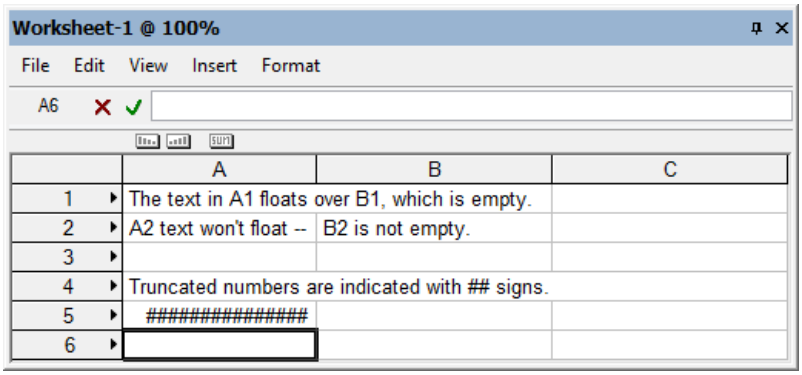
Formula Bar Display	Explanation
 No cell address displays; the current cell value is editable (Vectorworks Design Series required)	In database sub-rows, if Vectorworks Design Series products are installed, information that comes from the object's data record can be edited in the Formula bar, and the object's record is updated automatically. For example, the price data for a sub-row object could be updated. To accept the edits and change both the worksheet and record, click the green check mark.
 No cell address displays; a list of the values that are available for the current cell displays (Vectorworks Design Series required)	In database sub-rows, if Vectorworks Design Series products are installed, some fields that come from the object's data record can be edited, but they only allow certain pre-defined values. For example, the sill style for a window sub-row object could be changed in the Formula bar. Select the new value from the list to change both the worksheet and record.
 Neither a cell address nor a cell value displays	In database sub-rows, if the Vectorworks Architect or Vectorworks Landmark product is not installed, information that comes from the object's data record cannot be edited in the worksheet. To edit a value that displays in a sub-row, right-click (Windows) or Ctrl-click (Mac) the number of the row that is associated with the item in the drawing, and select the **Select Item** command from the context menu. Then use the Data tab of the Object Info palette to edit the object data as needed.

Resizing Rows and Columns

To adjust column width or row height, drag the divider bar between the column letters or row numbers. If you select multiple rows or columns before you drag the bar, all rows or columns are adjusted to the same size.

Alternatively, select the **Column Width** command from either the **Format** menu or the column context menu (see "Column Width" on page 1326). Adjust the row height with the **Row Height** command from either the **Format** menu or the row context menu (see "Row Height" on page 1326).

Text that is longer than the width of a cell "floats" over empty adjacent cells. Numbers that exceed the cell width are displayed with # characters. Alternatively, text can be set to wrap (see "Formatting Worksheet Cells" on page 1330).



To hide a row or column, set the **Column Width** or **Row Height** to 0. To display the column or row again, position the cursor over the column letter or row number adjacent to the hidden column or row. When the cursor changes to a double bar resize cursor, click-drag the bar to resize the column or row.

Cutting, Copying, and Pasting Cell Contents

The standard shortcut keys for Cut and Paste can be used for worksheet editing. The same value or formula can be copied and pasted to a range of cells.

To copy cell contents to a cell or range of cells:

1. Select the cell with the information to repeat, and then press Ctrl+C (Windows) or Command+C (Mac) to copy the cell.

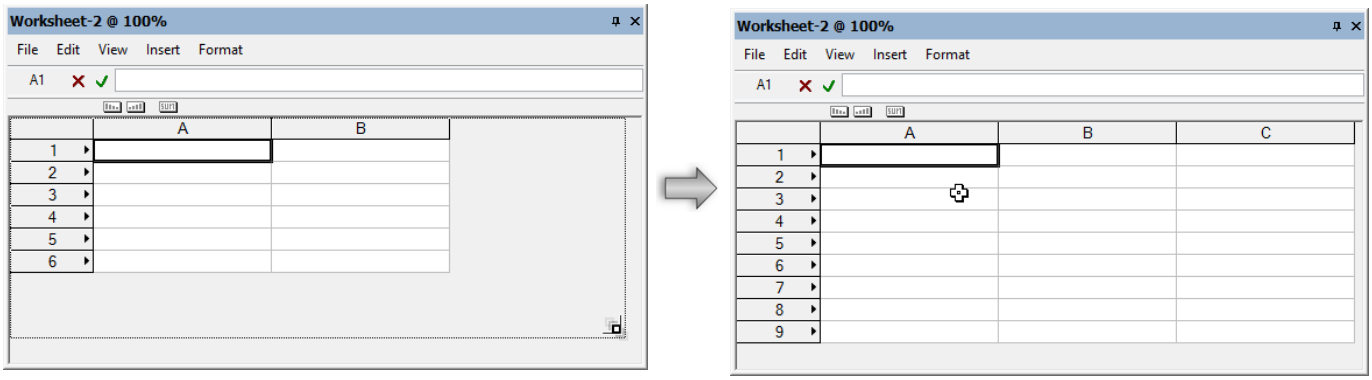
   Alternatively, select the **Copy** command from either the **Edit** menu or the appropriate context menu (see "Copy" on page 1325).

2. Select the destination cell(s) for the information, and then press Ctrl+V (Windows) or Command+V (Mac) to paste. The formula or value is repeated in each of the selected cells.

   Alternatively, select the **Paste** command from either the **Edit** menu or the appropriate context menu (see "Paste" on page 1325).

Adding Rows and Columns

From the **Insert** menu, you can insert rows and columns (see "Worksheet Commands" on page 1324). An empty row is added above the current row, or an empty column is added to the left of the current column. Another option is to position the cursor at the bottom right corner of the worksheet to activate a special resize cursor; drag as needed to add rows and columns to the bottom and right side of the worksheet.



Position the cursor over the bottom right corner of the worksheet to obtain the resize cursor; drag to add columns and/or rows

Moving Rows and Columns

Use the drag and drop method to move contiguous rows and columns or to move a copy of contiguous rows and columns.

To move a copy of rows or columns:

1. Click the column letter or row number to select a column or row (click-drag across the letters or numbers to select multiple columns or rows).
2. Press and hold Ctrl (Windows) or Option (Mac) and move the cursor to the edge of the selected rows or columns. When the cursor changes to a copy cursor to indicate that moving a copy is permitted, drag the selection to the desired location in the worksheet.

To move rows or columns:

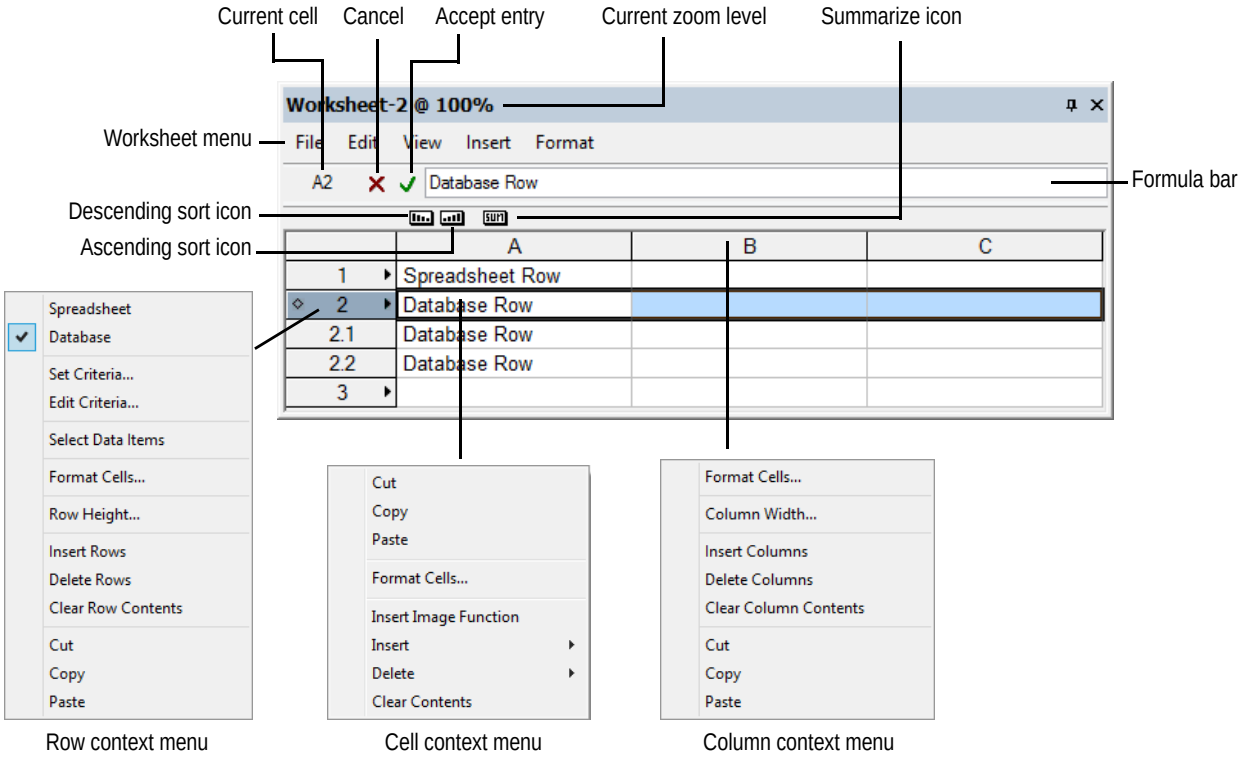
1. Click the column letter or row number to select a column or row (click-drag across the letters or numbers to select multiple columns or rows).

2. Move the cursor to the edge of the selected rows or columns. When the cursor changes to a move cursor to indicate that moving the rows or columns is permitted, drag the selection to the desired location in the worksheet.

Editing Record Information with Worksheets

Worksheet Commands

Various command menus are available in the worksheet window, as well as sorting functions. The main worksheet menu is at the top of the window. Right-click (Windows) or Ctrl-click (Mac) on a specific worksheet row or cell to open a context menu. To sort the sub-rows associated with a database header row, click and drag a sort icon to the column header cell.



Worksheet Menu
Worksheet Cell Context Menu
Worksheet Row Context Menu
Worksheet Column Context Menu

Worksheet Menu

The main **Worksheet** menu commands are described in the following table.

Worksheet Command	Description
File menu	
Recalculate	Recalculates all formulas in all worksheets, whether open or closed. This function can also be accessed from the context menu of the worksheet image (on the drawing): right-click (Windows) or Ctrl-click (Mac) on the worksheet, and select **Recalculate**.

Worksheet Command	Description
Preferences	Opens the Worksheet Preferences dialog box. **Header** and **Footer** text fields and the **Margin** settings apply to printed worksheets only. Select **Show Grid** to display the worksheet gridlines. Select **Show Tabs** to print worksheet column and row headers. Select **Auto-recalc** to recalculate all worksheet arithmetic functions when cells are edited. Click **Font** to specify the worksheet default font and size.
Printer Setup	Opens the Printer Setup dialog box. This is the same as the standard Printer Setup dialog box; however, it only affects the printer information for the worksheet.
Print	Opens the Print dialog box, to print the current worksheet; this is the only way to print a worksheet unless the worksheet is included as a part of the drawing
Edit menu	
Undo	Undoes the last worksheet change; execute the command multiple times to undo multiple actions
Redo	Reverses the last **Undo** command; execute the command multiple times to redo multiple undo actions
Cut	Removes the contents of selected cells, temporarily storing the contents in the clipboard
Copy	Copies the contents of selected cells to the clipboard, where they are temporarily stored; the original contents remain in the worksheet
Paste	Places cell contents stored in the clipboard into the current cell or range of cells
Clear Contents	Deletes the contents of the selected cells
Delete Rows	Deletes the selected row(s) from the worksheet. Use caution when deleting a row. Deleting cells that are part of a formula may change the values returned by the formula.
Delete Columns	Deletes the selected column(s) from the worksheet. Use caution when deleting a column. Deleting cells that are part of a formula may change the values returned by the formula.
View menu	
Database Headers	Toggles between displaying and hiding all worksheet database header rows
Grid Lines	Toggles between displaying and hiding grid lines between the rows and columns of the worksheet, in both the worksheet window and the worksheet image (on the drawing)
Zoom	Increases or decreases the zoom percentage by preset levels from 50% to 300%; the current zoom level displays in the worksheet title bar. Select a zoom level from the **Worksheet** menu, or roll the mouse wheel while holding Ctrl (Windows) or Option (Mac) to increase or decrease the zoom level by increments of 10% (regardless of the number of lines you assigned the mouse to scroll in the mouse setup). This feature will not work properly if standard scrolling is disabled in the mouse setup. For example, if the mouse's scrolling size is set to "none," mouse zooming in the Vectorworks program is disabled. (The specific settings required for this feature depend on the type of mouse being used.)
Insert menu	

Worksheet Command	Description
Rows	Adds rows to the worksheet, above the selected row(s). The number inserted depends on how many rows in the worksheet are highlighted at the time the command is selected. Use caution when inserting rows. Depending on the type of cell references used in formulas, inserting rows could change the values returned by a formula.
Columns	Adds columns to the worksheet, to the left of the selected column(s). The number inserted depends on how many columns in the worksheet are highlighted at the time the command is selected. Use caution when inserting columns. Depending on the type of cell references used in formulas, inserting columns could change the values returned by a formula.
Function	Opens the Select Function dialog box; select a function to be inserted in the formula (see “Entering Formulas in Worksheet Cells” on page 1335)
Criteria	Opens the Criteria dialog box; select search criteria to insert in a formula
Image Function (Vectorworks Design Series required)	Inserts the image function in the formula for the current cell
Format menu	
Cells	Opens the Format Cells dialog box, for setting the format and appearance of selected cells
Column Width	Opens the Column Width dialog box. Set the width value of selected cells in the specified units. Click **Standard Width** to use the default width. The width of multiple selected columns can be adjusted at one time.
Row Height	Opens the Row Height dialog box; set the row height to automatically fit the selected cell contents, or set a specific row height in the specified units. The height of multiple selected rows can be adjusted at one time.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Worksheet Cell Context Menu
Worksheet Row Context Menu
Worksheet Column Context Menu

Worksheet Cell Context Menu

To access the commands available for a specific worksheet cell, right-click (Windows) or Ctrl-click (Mac) on the cell.

Menu Item	Description
Cut	Removes the contents of selected cells, temporarily storing the contents in the clipboard
Copy	Copies the contents of selected cells to the clipboard, where they are temporarily stored; the original contents remain in the worksheet
Paste	Places cell contents stored in the clipboard into the current cell or range of cells
Format Cells	Opens the Format Cells dialog box, for setting the format and appearance of selected cells
Insert Image Function (Vectorworks Design Series required)	Inserts the image function in the formula for the current cell
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Menu Item	Description
Insert	Adds rows or columns to the worksheet. The number inserted depends on how many rows or columns in the worksheet are highlighted at the time the command is selected. Select **Insert > Rows** to insert above the selected row(s). Select **Insert > Columns** to insert to the left of the selected column(s). Use caution when inserting rows or columns. Depending on the type of cell references used in formulas, inserting rows or columns could change the values returned by a formula.
Delete	Deletes rows or columns from the worksheet. Select one or more rows or columns and select **Delete > Rows** or **Delete > Columns**. Use caution when deleting a row or column. Deleting cells that are part of a formula may change the values returned by the formula. Select **Edit > Undo** to undo the action.
Clear Contents	Deletes the contents of the selected cells
Pick Value from List (Vectorworks Design Series required)	If the cell is in a database sub-row, and the column lists a field that only allows certain pre-defined values, use this option to edit the object's data. For example, you might want to change the sill style for several window objects from the window schedule. Select the Sill cells for the objects to be changed, and right-click (Windows) or Ctrl-click (Mac). Select a different sill type from the list of options to change both the worksheet and the objects' data records.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Worksheet Menu
Worksheet Row Context Menu
Worksheet Column Context Menu

Worksheet Row Context Menu

To access the commands available for a specific worksheet spreadsheet or database header row, right-click (Windows) or Ctrl-click (Mac) while on the row number. These commands do not apply to database sub-rows.

Menu Item	Description
Spreadsheet	Converts a database header row into a row of spreadsheet cells. This deletes all sub-rows and the information contained within them. Any formulas that were defined in the columns of the header row remain intact. This command has no effect on spreadsheet cells.
Database	Converts a row of spreadsheet cells into a database header row and opens the Criteria dialog box. This command has no effect on database rows.
Set Criteria	Opens the Criteria dialog box for setting the criteria that is used to generate the database sub-rows. Available only when a database header row is clicked.
Edit Criteria	Opens the Criteria dialog box for editing the criteria that is used to generate the database sub-rows. Available only when a database header row is clicked.
Select Data Items	Selects all objects on the drawing that meet the criteria for the database row. Available only when a database header row is clicked.
Format Cells	Opens the Format Cells dialog box, for setting the format and appearance of selected cells
Row Height	Opens the Row Height dialog box; set the row height to automatically fit the selected cell contents, or set a specific row height in the specified units. The height of multiple selected rows can be adjusted at one time.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Menu Item	Description
Insert Rows	Adds rows to the worksheet, above the selected row(s). The number inserted depends on how many rows in the worksheet are highlighted at the time the command is selected. Use caution when inserting rows. Depending on the type of cell references used in formulas, inserting rows could change the values returned by a formula.
Delete Rows	Deletes the selected row(s) from the worksheet. Use caution when deleting a row. Deleting cells that are part of a formula may change the values returned by the formula.
Clear Row Contents	Deletes the contents of the selected cells
Cut	Removes the contents of selected cells, temporarily storing the contents in the clipboard
Copy	Copies the contents of selected cells to the clipboard, where they are temporarily stored; the original contents remain in the worksheet
Paste	Places cell contents stored in the clipboard into the current cell or range of cells

Database Sub-Row Context Menu

While on a database sub-row, right-click (Windows) or Ctrl-click (Mac) and select the **Select Item** command from the context menu. Use this command to select an individual database object in the drawing; the view changes to display the selected object (see “Importing Worksheets” on page 1358). The command is unavailable if the sub-row is summarized (see “Database Row Sort and Summary Functions” on page 1329).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Worksheet Menu
Worksheet Cell Context Menu
Worksheet Column Context Menu

Worksheet Column Context Menu

To access the commands available for a specific worksheet column, right-click (Windows) or Ctrl-click (Mac) while on the column letter.

Menu Item	Description
Format Cells	Opens the Format Cells dialog box, for setting the format and appearance of selected cells
Column Width	Opens the Column Width dialog box. Set the width value of selected cells in the specified units. Click **Standard Width** to use the default width. The width of multiple selected columns can be adjusted at one time.
Insert Columns	Adds columns to the worksheet, left of the selected column. The number inserted depends on how many columns in the worksheet are highlighted at the time the command is selected. Use caution when inserting columns. Depending on the type of cell references used in formulas, inserting columns could change the values returned by a formula.
Delete Columns	Deletes the selected columns from the worksheet. Use caution when deleting a column. Deleting cells that are part of a formula may change the values returned by the formula.
Clear Column Contents	Deletes the contents of the selected cells
Cut	Removes the contents of selected cells, temporarily storing the contents in the clipboard
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Menu Item	Description
Copy	Copies the contents of selected cells to the clipboard, where they are temporarily stored; the original contents remain in the worksheet
Paste	Places cell contents stored in the clipboard into the current cell or range of cells

Worksheet Menu
Worksheet Cell Context Menu
Worksheet Row Context Menu

Database Row Sort and Summary Functions

The group of sub-rows associated with a header row can be sorted and summarized in different ways. These functions can be set independently for each database header row in a worksheet.

The sort and summary functions cannot be used on a header cell for an image column (Vectorworks Design Series required).

To sort or summarize a group of database sub-rows:

1. If the database header rows are not displayed, select **View > Database Headers** from the **Worksheet** menu.
2. Select the header row of the group of sub-rows to sort or summarize; the header row has a diamond next to its number.

   The three icons above the left end of the column header cells become available.

Icon Item	Description
Descending Sort 	Sorts the database sub-rows in descending order, according to the contents of this column
Ascending Sort 	Sorts the database sub-rows in ascending order, according to the contents of this column
Summarize 	Summarizes the database sub-rows according to the contents of this column. Sub-rows that have identical items in this column are grouped together in a single row. If a column contains data from a numeric field, the summarized column contains a sum of the values for all objects that are grouped on the row. This may be appropriate for some columns, but not others. For example, you might have a window schedule that sorts and summarizes the data by the Window ID column. You would want the Quantity column to show the sum of all windows with a particular ID, but you would want the Window Height column to show the height of a single window with that ID (not the height of all windows combined). Add an additional summary operator to the Window Height column to show the correct numeric data.

3. Click and drag an icon to the column header cell to be used for the sort or summary. A new icon displays next to the column heading letter. For an ascending or descending sort, a number in the icon indicates the sort precedence for that column.
4. Apply additional sort or summary icons as needed. In each group of sub-rows, up to 20 columns can have either an Ascending or Descending Sort icon, and any number of columns can have a Summarize icon. The Summarize icon can be used on a column by itself, or in conjunction with one of the sort icons.
5. To remove a sort or summary, click and drag the icon away from the column header cell.

Formatting Worksheet Cells

The appearance of worksheet cells can be set by a variety of formatting options.

Formatting applied to a database header row applies to all of the associated database sub-rows.

To format worksheet cells:

1. Select the cell(s) to format.
2. From the **Worksheet** menu, select **Format > Cells**.

   The Format Cells dialog box opens.

   On the Number tab, set the number format for the selected cells.

Click to show/hide the parameters.

Parameter	Description
General	Specifies the default general format
Decimal	Uses decimal numbers; enter a value for the number of decimal places, and if desired, select to use commas as separators
Scientific	Uses scientific numbers; enter a value for the number of decimal places
Fractional	Uses fractional numbers; enter the rounding value for fractions
Percentage	Uses percentages; enter a value for the number of decimal places
Dimension	Uses dimension numbers
Dimension Area	Uses the dimension area format (precision and units) as specified for this document; also displays the area units after the number
Dimension Volume	Uses the dimension volume format (precision and units) as specified for this document; also displays the volume units after the number
Angle	Determines the accuracy of angles and the measurement system used; the measurement system can be degrees/minutes/seconds, or decimal numbers up to eight decimal places
Date	Uses date formats; select the desired format from the list
Boolean	Select to evaluate the cell data to either True or False
Text	Select to enter a string of characters; the cell contents are treated as text, even if a number is in the string
Leader	Displays the specified leader text before the cell value (except for Boolean and Text formats)
Trailer	Displays the specified trailer text after the cell value (except for Boolean and Text formats)

3. Click the Alignment tab to specify text alignment options.

Click to show/hide the parameters.

Parameter	Description
Alignment	Sets the alignment of text in relation to the cell border
Horizontal	Specifies horizontal text alignment; select General to align text strings on the left and numbers on the right
Vertical	Specifies vertical text alignment
Text Orientation	Sets the text orientation

Parameter	Description
Vertical Text	Orients text vertically
Horizontal Text	Orients text horizontally
Wrap Text	Select to wrap text that exceeds the cell width (automatically adjusting row height); deselect to allow text that is longer than the cell width to "float" over empty adjacent cells. If adjacent cells contain content, unwrapped text may appear truncated. Numbers that exceed the cell width are displayed with # characters.
Merge Cells	Merges a range of selected spreadsheet cells into one cell; cell and border formatting and text wrapping are applied to the cell group rather than to the individual cells. The cell contents and format of only the upper left cell in the group apply to the merged cells. Data and formatting in the other cells will be lost during the merge. To split merged cells, select the merged cell group and then deselect **Merge Cells**.

4. Click the Font tab to specify the font, font size, style, and color of text in selected cells. See "Formatting Text" on page 389.
5. Click the Border tab to set cell border formatting options.

   Select the Line Attributes, and then use the Presets or Preview buttons to add or remove border elements.

Click to show/hide the parameters.

Parameter	Description
Line Attributes	
Style	Sets the line style and thickness for the border element
Color	Sets the line color for the border element
Presets	
None	Removes all border formatting
Outline	Adds or removes a border only on the outside edges (top, bottom, left, and right) of the cell selection
Inside	Adds or removes a border only on the inside (horizontal and vertical) edges of the cell selection
Preview	
Top	Adds or removes a border at the top of the cell selection
Horizontal	When more than one cell is selected, adds or removes a horizontal border between cells
Bottom	Adds or removes a border at the bottom of the cell selection
Left	Adds or removes a border on the left edge of the cell selection
Vertical	When more than one cell is selected, adds or removes a vertical border between cells
Right	Adds or removes a border on the right edge of the cell selection

6. Click the Patterns tab to specify fill options for the selected cell(s).

Click to show/hide the parameters.

Parameter	Description
Style	Select None to remove current cell fill options or to specify no fill; select Solid or Pattern to apply a solid fill color or pattern to the selected cell(s)
Color/Pattern	If **Style** is set to Solid, select the fill **Color**; if **Style** is set to Pattern, select the **Pattern** and the foreground and background colors

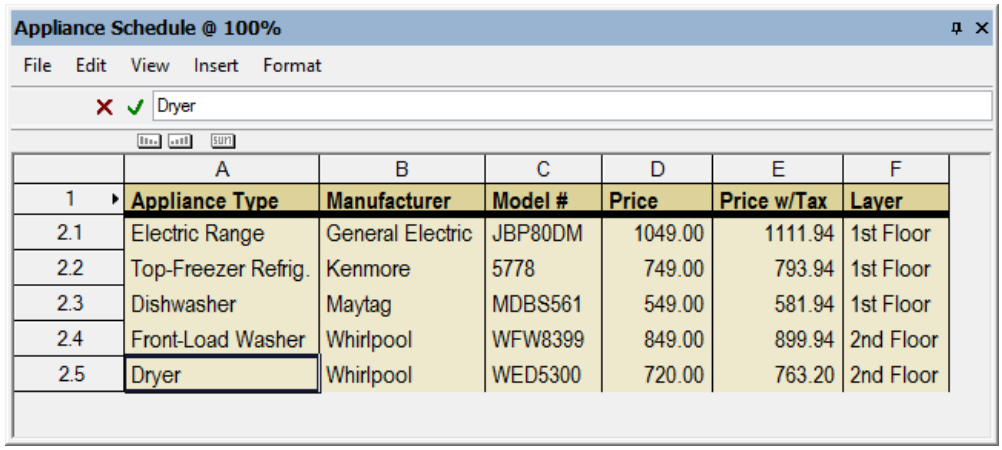
7. If Vectorworks Design Series is installed, click the Images tab to specify the type, size, view, and margin for images in the selected cells. For more information, see “Inserting Images in Worksheets” on page 1343.

Click to show/hide the parameters.

Parameter	Description
Image Type	Select Thumbnail to display a thumbnail image of the referenced object in the cell; this is useful for 2D/3D symbols or plug-in objects. Use other settings to control the scale, view, rotation, and render mode shown in the thumbnail. Select 2D Attributes to display a rectangle with the same 2D attributes as the referenced object. For a styled wall, slab, or space, the rectangle fill is a top/plan cross-section view of the object.
Size	
AutoSize	Sizes the image automatically so that the entire image is always visible, even when the cell size changes
Fixed	Sizes the image according to the specified **Height** and **Width**; also specify the **Units** for the dimensions (points, millimeters, or inches)
Layer Scale	Sizes the image according to the scale of the layer on which the object appears (thumbnails only)
Custom Scale	Select this option and then click **Scale** to select or enter a scale for the image (thumbnails only)
View	
Standard Views	Select from any of the standard Vectorworks views for the image (thumbnails only)
Rotation	If Top/Plan view is selected, enter the number of degrees to rotate the image (thumbnails only)
Render Mode	For any view except Top/Plan, select the render mode for the image (Wireframe, OpenGL, or Hidden Line); Top/Plan view is always rendered in Wireframe mode. Applies to thumbnails only.
Margin	
Units	Enter the units to use for the **Size** of the margin around the image (points, millimeters, or inches)
Size	Enter the size of the margin around the image, in the specified **Units**

8. Click **OK** to set the formatting for the selected cell(s). The worksheet formatting also applies to worksheets placed on a drawing.

Formatted worksheet in the worksheet window



	A	B	C	D	E	F
1	Appliance Type	Manufacturer	Model #	Price	Price w/Tax	Layer
2.1	Electric Range	General Electric	JBP80DM	1049.00	1111.94	1st Floor
2.2	Top-Freezer Refrig.	Kenmore	5778	749.00	793.94	1st Floor
2.3	Dishwasher	Maytag	MDBS561	549.00	581.94	1st Floor
2.4	Front-Load Washer	Whirlpool	WFW8399	849.00	899.94	2nd Floor
2.5	Dryer	Whirlpool	WED5300	720.00	763.20	2nd Floor

Formatted worksheet placed on the drawing

Appliance Type	Manufacturer	Model #	Price	Price w/Tax	Layer
Electric Range	General Electric	JBP80DM	1049.00	1111.94	1st Floor
Top-Freezer Refrig.	Kenmore	5778	1600.00	1696.00	1st Floor
Dishwasher	Maytag	MDBS561	549.00	581.94	1st Floor
Front-Load Washer	Whirlpool	WFW8399	849.00	899.94	2nd Floor
Dryer	Whirlpool	WED5300	510.00	540.60	2nd Floor

Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Attributes Palette

Entering Data in Spreadsheet Cells

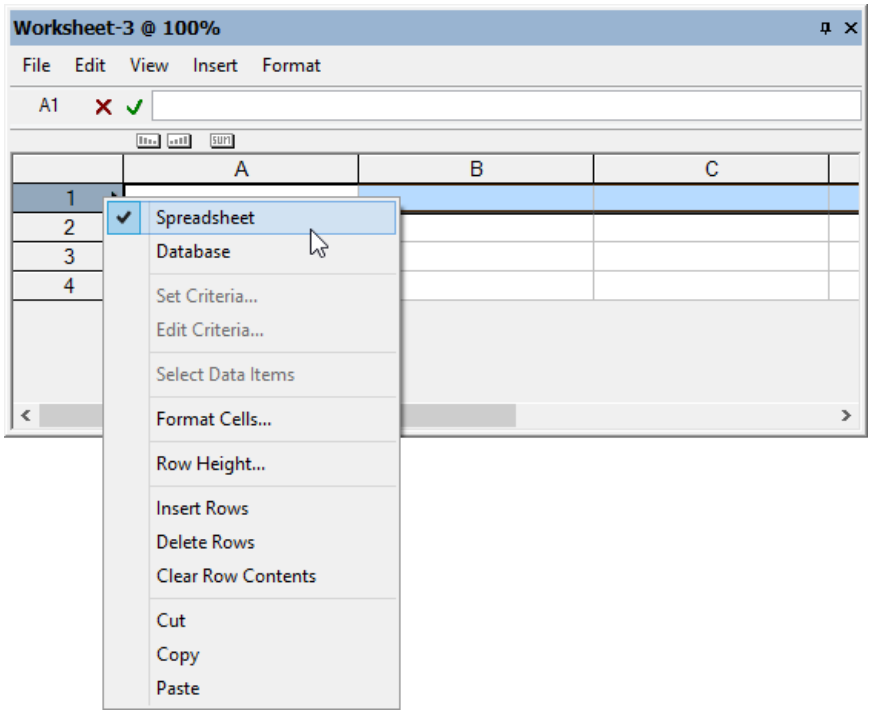
Three types of information can be entered into the spreadsheet cells of a worksheet: constant values (including text or numbers), formulas, and images (Vectorworks Design Series required). In addition, a cell can reference another cell in the same worksheet or in another worksheet.

- Text helps to identify the purpose of a worksheet and labels the columns in a worksheet.
- Images add visual information about items on a worksheet, and can also be used to create a drawing legend (Vectorworks Design Series required).
- Use formulas to perform calculations based on drawing data. A formula can be a simple mathematical equation, or it can include one or more built-in functions. The Vectorworks program provides mathematical functions (for example, a square root function), as well as functions that pull information from drawing objects (for example, a function that returns the volume of selected objects). See “Worksheet Functions” on page 1344 for a list of the functions available.

  Database record fields that are attached to objects in the drawing (such as Model Number or Price) cannot be used in a spreadsheet cell. To include this type of data in the worksheet, see “Entering Data in Database Rows” on page 1339.

To define a spreadsheet row:

1. Right-click (Windows) or Ctrl-click (Mac) on the number of the row to change.
2. From the **Row** context menu, select **Spreadsheet**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



3. The cells in the row are empty until you define the contents. Select a cell, and then enter the desired information in the worksheet Formula bar located at the top of the worksheet.
   - To enter text or numbers, see “Entering Constant Values in Worksheet Cells” on page 1334.
   - To enter a formula, see “Entering Formulas in Worksheet Cells” on page 1335.
   - To reference other cells in this cell, see “Referencing Other Worksheet Cells” on page 1338.
   - To insert an image, see “Inserting Images in Worksheets” on page 1343.

Entering Constant Values in Worksheet Cells

Constant values consist of numbers, spaces, non-numeric characters, or any combination of these. Constant values are not part of a formula or the result of a formula.

The formula phrase “=1”, or any number following an equal sign, is also considered a constant value.

Select the cell, and then enter the text or numbers needed; your entries automatically display in the worksheet Formula bar. When you click the green check mark on the Formula bar, the value displays in the cell.

Keep in mind the following:

- Most constant values are treated as text and default to the General format. However, some combinations of numeric and non-numeric characters are interpreted as a particular number format. For example, an entry of 07/19/2013 automatically changes the format to the month/day/year date format. (See “Formatting Worksheet Cells” on page 1330.)
- Text is left-aligned unless the cell is formatted differently on the Alignment tab in the Format Cells dialog box (select **Format > Cells** from the **Worksheet** menu).
- Numbers entered in cells default to the General format. To change them to another format (for example, dimension or fractional), select **Format > Cells** from the **Worksheet** menu, and select the new format on the Number tab in the Format Cells dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Entering Data in Spreadsheet Cells
Entering Data in Database Rows
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Entering Formulas in Worksheet Cells

Use formulas to evaluate and perform operations on drawing data. Formulas always begin with an equal sign (=) and consist of a combination of functions, cell references, or operators that combine values to produce a new value.

Formulas must be entered with a specific syntax. If the formula is not entered correctly, the formula entry itself displays in the cell, instead of the result of the formula. Two common mistakes in syntax include forgetting to use pairs of parentheses, and omitting required commas when no argument is present. Formula syntax is described in the following table.

	Symbol	Explanation	Example
General Syntax	Equal sign =	Begins each formula; also indicates a value for a variable	`=CriteriaVolume(t=wall)`
	Parentheses `( )`	Encloses a function argument; also used in arithmetic equations	`=acos(0.6)` `=A6+(A6*.07)`
	Square brackets `[ ]`	Encloses a record destination	`=R IN ['myformat']`
	Period .	Separates a record identifier and a field identifier	`=Furniture.Type`
	Colon :	Separates path name levels in cell references	`=MyWorksheet:A1`
	Comma or semicolon , or ;	Separates multiple values in a function argument; use a semicolon when commas are used as decimal separators by the operating system	`=sum(A2,E3)` `=sum(A2;E3)`
	Single quote '	Encloses a string constant	`=Appliances.'Model #'`
	Dollar sign $	Designates an absolute reference	`=A4*$B$1`
	Double period . .	Designates a range of cells	`=sum(A10..A12)`
Arithmetic Operators	Plus sign +	Addition	`=A6+A8`
	Hyphen -	Subtraction	`=A6-A8`
	Asterisk *	Multiplication	`=A6*.06`
	Forward slash /	Division	`=B3/12`
	Caret ^	Exponentiation	`=13^2`
	DIV	Integer division (returns the integer quotient of the division operation)	`j:= 36 DIV 5;`
	MOD	Remainder division (returns the remainder of the division operation as an integer)	`k:= 36 MOD 5;`

	Symbol	Explanation	Example
Comparison Operators (used with IF function)	Equal sign =	Equal	`=if((L='L2'),Area,0)`
	Less than and greater than signs (or Option+ = on Mac) <> or ≠	Not equal	`=if((S<>'Dryer'),B9,0)`
	Less than sign <	Less than	`=if((C7<100),100,C7)`
	Less than and equal signs (or Option+ < on Mac) <= or ≤	Less than or equal to	`=if((E2<=G2),0.05,G2)`
	Greater than sign >	Greater than	`=if((C7>100),100,C7)`
	Greater than and equal signs (or Option+ > on Mac) >= or ≥	Greater than or equal to	`=if((E2>=G2),0.05,G2)`

To force the program to treat a number as text, enclose the number in single quotation marks, as in '40'; or format the cell as **Text** on the Number tab of the Format Cells dialog box.

Formulas follow standard algebraic rules of hierarchy. In the following example, the value in cell C28 is first multiplied by 12, and then 4.5 is subtracted from that value. The result is then divided by 12.

```
=((C28*12)-4.5)/12
```

There are several built-in functions that can be used in formulas, including mathematical functions and functions that pull information from objects in the drawing. To use one or more worksheet functions in a formula, either enter the function manually, or use the **Insert > Function** and **Insert > Criteria** commands to select a function and selection criteria (if required) from dialog boxes. See “Worksheet Functions” on page 1344 for more information about how to use functions.

If there is a logic problem or calculation error with a formula, an error code displays in the cell.

Error Code	Explanation
#NAME?	The field name in a database header row definition does not exist in the file; see “Retrieving Record Information in a Worksheet” on page 1342
#VALUE!	The argument is the wrong type of data for the formula; for example, a cell referenced in a mathematical formula contains text
#CVAL?	A cyclical reference cannot be resolved
?Result?	The result value type is unrecognized
#OPCODE?	Internal error
#DIV 0!	Division by zero error encountered
#FAC?	Unrecognized entry
#OBJ!	The worksheet name in an external reference does not exist, or the record name does not exist
#CSTATUS?	Functions are nested too deeply (more than ten levels deep)

To manually enter a formula:

1. Select the cell.

2. Enter an equal sign (=), and then enter the formula. The entries automatically display in the worksheet Formula bar. A formula can consist of functions, operators, cell references, and constant values.
3. When the formula is complete, click the green check mark or press Enter to validate the entry. To cancel an entry, click the red X or press Esc.
4. The formula executes as soon as the cell entry has been validated (**Auto-recalc** must be selected in the worksheet preferences; see "Preferences" on page 1325).

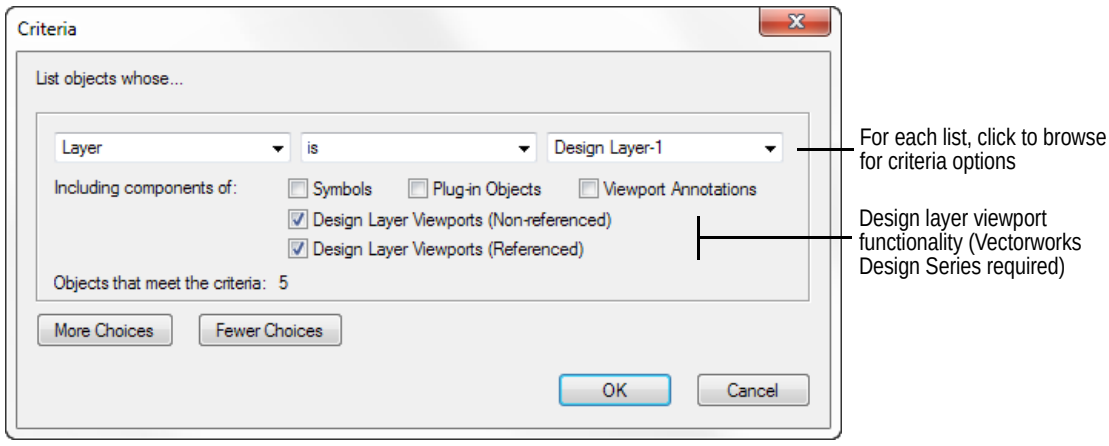
To enter a formula with the **Function** and **Criteria** commands:

1. Select the cell.
2. Enter an equal sign (=). The entry automatically displays in the worksheet Formula bar.
3. Select **Insert > Function** from the **Worksheet** menu.

   The Select Function dialog box opens.
4. Select a function from the list and click **OK**.

   The formula is placed in the worksheet Formula bar and the cursor is placed between the parentheses, awaiting an argument, if necessary.
5. Do one of the following:
   - If the function requires numbers or text, enter the argument between the parentheses and proceed to step 8.
   - If the function requires selection criteria, select **Insert > Criteria** from the **Worksheet** menu.
6. If an object is selected when the **Criteria** command is selected, the Paste Attributes dialog box opens. Otherwise, proceed to step 7.

   Do one of the following:
   - To use attributes of the selected object as the only selection criteria, select the attributes and click **OK**. Proceed to step 8.
   - To specify other criteria, or to use attributes of other objects in the drawing, click the **Custom** button.
7. The Criteria dialog box opens. Set each of the three fields to the desired selection criteria. Click **More Choices** to specify additional criteria. Click **Fewer Choices** to remove added criteria. Click **OK** to add the criteria to the function argument.



8. When the formula is complete, click the green check mark or press Enter to validate the entry. To cancel the entry, click the red X or press Esc.
9. The formula executes as soon as the cell entry has been validated (**Auto-recalc** must be selected in the worksheet preferences; see "Preferences" on page 1325).

Entering Data in Spreadsheet Cells
Entering Data in Database Rows
Entering Constant Values in Worksheet Cells
Referencing Other Worksheet Cells
Worksheet Functions

Referencing Other Worksheet Cells

A formula can reference the contents of one or more other cells. The cells can be referenced within the current worksheet (internal references), or from another worksheet (external references) within the same file.

External references must include the full path name to the other worksheet. The following table shows the syntax for entering an external reference into a formula.

Syntax	Example
worksheet name:cell address	`=MyWorksheet:A1`
worksheet name:range of addresses	`=SUM(MyWorksheet:A1..A12)`

If the name of the worksheet contains spaces, the name must be enclosed with single quotes as in the following example: ='Appliance Schedule':A1

To update an external reference, select **File > Recalculate** from the **Worksheet** menu.

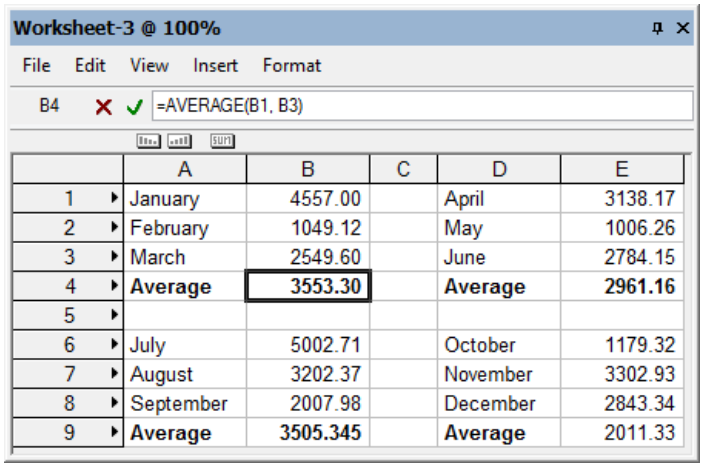
Relative and Absolute Cell References

Cell references in a worksheet can be either relative and absolute. When the formula that contains the reference is moved, an absolute reference always refers to the original cell address, while a relative reference changes depending on the location of the cell that contains the reference.

Use the dollar sign ($) character to indicate an absolute reference. The $ character locks the part of the cell reference it precedes, as described in the following table.

Combination	Description
$A1	Locks the specified column reference but leaves the row reference relative; the same column is always referred to, but the row changes if the formula is placed in a different row
A1	Locks both the specified column and row references; regardless of where the formula is copied, it always refers to the original cell
A$1	Locks the specified row reference but leaves the column reference relative; the same row is always referred to, but the column changes if the formula is placed in a different column

In the following example worksheet, the formula =AVERAGE(B1..B3) is in cell B4. If the formula were copied to cell E9, the formula would automatically be changed to =AVERAGE(E6..E8). Because the references are relative, both the column and row would change relative to the cell where the formula is placed—always indicating the three cells directly above the formula.



	A	B	C	D	E
1	January	4557.00		April	3138.17
2	February	1049.12		May	1006.26
3	March	2549.60		June	2784.15
4	**Average**	**3553.30**		**Average**	**2961.16**
5					
6	July	5002.71		October	1179.32
7	August	3202.37		November	3302.93
8	September	2007.98		December	2843.34
9	**Average**	**3505.345**		**Average**	2011.33

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Entering Data in Spreadsheet Cells
Entering Data in Database Rows
Entering Formulas in Worksheet Cells
Entering Constant Values in Worksheet Cells

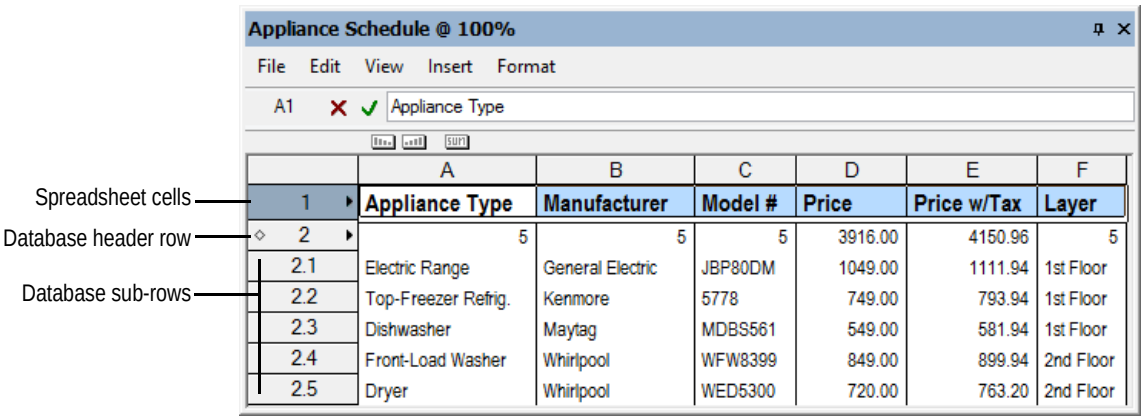
Entering Data in Database Rows

Database rows display data fields, calculations, or images associated with the objects in a drawing. The database header row is identified by a diamond shape next to the row number. When you create the database row, set the criteria to determine which objects will be listed in the related sub-rows. For example, you might set a header row to list all symbols in the drawing. A sub-row would then be generated for each symbol in the drawing. (If no object meets the header row criteria, no sub-rows are created.)

Many criteria combinations can be specified, such as class, object type, record information, or line weight. For example, create a list of all the rooms in a resort, or list only the green wing-backed chairs from all the two-room suites that are scattered throughout the resort.

In each column in the database header row, specify which information about the objects to display. A column can list a specific attribute of each sub-row object, such as its class or layer. A column can also list a data field contained in a record attached to each object. Alternatively, a column can contain a constant, an image, or a formula, just as a spreadsheet cell can.

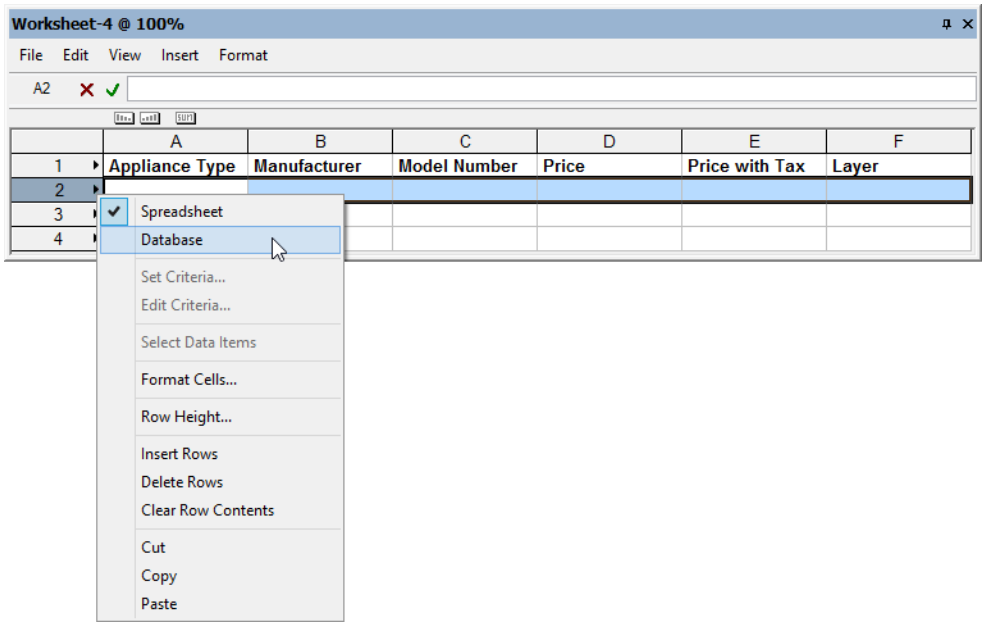
In the following example, database header row 2 has its criteria set to list all the objects in the drawing that have the appliance record attached to them. Columns A through D list the contents of the data fields in the appliance record: the appliance type, manufacturer, model number, and price. Column E contains a formula, which uses the value in column D to calculate the price of the appliance with sales tax. Column F lists which layer of the drawing contains the object.



	A	B	C	D	E	F
1	**Appliance Type**	**Manufacturer**	**Model #**	**Price**	**Price w/Tax**	**Layer**
◇ 2	5	5	5	3916.00	4150.96	5
2.1	Electric Range	General Electric	JBP80DM	1049.00	1111.94	1st Floor
2.2	Top-Freezer Refrig.	Kenmore	5778	749.00	793.94	1st Floor
2.3	Dishwasher	Maytag	MDBS561	549.00	581.94	1st Floor
2.4	Front-Load Washer	Whirlpool	WFW8399	849.00	899.94	2nd Floor
2.5	Dryer	Whirlpool	WED5300	720.00	763.20	2nd Floor
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To create a database row:

1. Right-click (Windows) or Ctrl-click (Mac) on the number of the row to change.

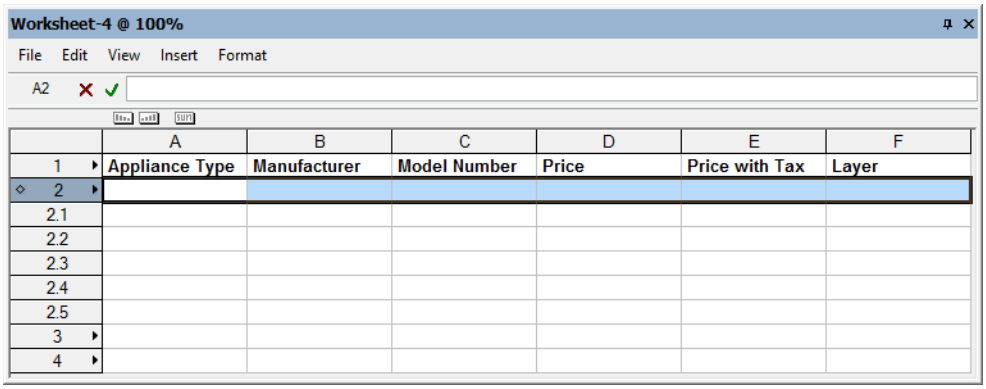


2. From the **Worksheet row** menu, select **Database**.

   The Criteria dialog box opens.

3. Specify the selection criteria for which objects to display in the sub-rows. The number of objects that meet the criteria displays, to help you verify that the criteria is correct. To specify additional criteria, click **More Choices**.

4. Click **OK** to enable database functionality for the row. Beneath the header row, sub-rows are created for each drawing object that meets the criteria specified. The columns are empty until you define which data from the objects to display in each column.



5. Select each database header cell, and specify the information to be shown in each column of the row:

   - To list attributes of each object (such as layer or class), see “Retrieving Object Attributes in a Worksheet” on page 1341.
   - To list record data associated with each object (such as color or price), see “Retrieving Record Information in a Worksheet” on page 1342.
   - To show the results of a formula for each object, see “Entering Formulas in Worksheet Cells” on page 1335.
   - To show an image for each object, select **Insert > Image Function** from the **Worksheet** menu. Alternatively, select **Insert Image Function** from the context menu. See “Inserting Images in Worksheets” on page 1343.

6. Each sub-row cell displays the information requested. Each cell in the header row displays the total number of objects found, or, if the column returns numerical data, the header cell displays the sum for all sub-rows. Information found in each column can be sorted using the ascending, descending, and summarize buttons; see “Database Row Sort and Summary Functions” on page 1329.

Undefining a Database Row

To undefine a database row:

1. Right-click (Windows) or Ctrl-click (Mac) on the number of the database header row to change.
2. Select **Spreadsheet**.

   Undefining a database row removes the database row criteria and all sub-rows. Any formulas that were defined in the columns of the header row remain intact.

Retrieving Object Attributes in a Worksheet

A drawing object can have several attributes, such as the layer it is on, the type of object it is, the symbol name (if it is a symbol), and whether it is currently selected. You can display this information in the database rows of a worksheet.

To retrieve object information in the database rows:

1. Click the cell in the database header row where the formula will be entered.
2. Enter an equal sign (=), and then enter the criteria to display. For example, enter =C to display the name of the class to which each object belongs. The entries display in the worksheet Formula bar.

Code	Criteria Name	Code	Criteria Name
ALL	Every object	PB	Pen background
AR	Arrowhead	PF	Pen Foreground
ASZ	Marker size	PLA	Plane
C	Class name	PON	Plug-in object name
FB	Fill background	PP	Pen pattern (same as LS)
FF	Fill foreground	R	Object record
FOT	Font	S	Symbol name
FP	Fill pattern	SEL*	Selection status
FSZ	Font size	SLST	Slab style
GFI	Gradient fill	SST	Sketch style
HFI	Hatch fill	ST	Object subtype
IFC_ENTITY	IFC entity	STO	Story
IFI	Image fill	T	Object type
L	Layer name	TFI	Tile fill
LS	Line style (same as PP)	TSTY	Text style
LT	Line type	TX	Texture
LW	Line weight	V	Visibility
N	Object name	VSEL*	Visible selection status
OPA	Opacity	WST	Wall style

Code	Criteria Name	Code	Criteria Name
* When used with the COUNT function, the SEL criterion counts objects that are actually non-selectable, such as the individual items within a group. The VSEL criterion counts only the visibly selected items, which is the same counting method used for the Object Info palette. For example, if you select and count a group that has 11 items in it, the SEL criterion returns a value of 12 (the group, plus the 11 items). The VSEL criterion returns a value of 1 (the group only).			

3. Click the green check mark to validate the entry.

Retrieving Record Information in a Worksheet

Database records are created in the Record Formats dialog box. These records are then assigned to objects through the Data tab of the Object Info palette. See "Viewing and Editing Object Records" on page 266 for more information. This information can be displayed in the database rows of a worksheet.

To retrieve record information in a database row:

1. Click the cell in the database header row where the formula will be entered.
2. Enter an equal sign (=), and then enter the record information to display. The entries display in the worksheet Formula bar. The syntax for retrieving record information is:

Syntax	Example
`=record name.field name`	`=Furniture.Type`

   A period (.) must separate the two names, or the formula will not be executed.

   If the name of the record format or field name contains spaces, the name must be enclosed with single quotes as in the following example: ='Appliance Record'.'Model Number'

3. Click the green check mark to validate the entry.

   The database information attached to each object displays in the sub-rows.

Selecting and Editing Database Objects

You can use the database rows in a worksheet to select the object(s) in the drawing that are related to that row. If Vectorworks Design Series is installed, you can also edit the information associated with many database objects from the worksheet.

To select database objects:

1. Either all database objects or a single database object can be selected.
   - To select all database objects that meet the database row criteria, right-click (Windows) or Ctrl-click (Mac) the row number of the database header row to open the context menu.
   - To select an individual database object, right-click (Windows) or Ctrl-click (Mac) the row number of the sub-row that contains the object to open the context menu.
2. From the context menu, select either **Select Data Items** or **Select Item**.

   All database objects that are represented by the header row, or the individual row object, are selected. If an individual object was selected with **Select Item**, the drawing view changes to display the selected object. The **Select Item** command is unavailable if the sub-row is summarized (see "Database Row Sort and Summary Functions" on page 1329).

D To edit database objects:

In database sub-rows, some information can be edited, but some cannot. For example, the results of a calculation cannot be edited. However, if Vectorworks Design Series is installed, the data associated with a database object can be edited in the worksheet, and the object's data record will be updated as well. For details, see "Editing Cell Contents" on page 1321.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Entering Constant Values in Worksheet Cells
Entering Formulas in Worksheet Cells
Worksheet Functions
Inserting Images in Worksheets

Inserting Images in Worksheets

If Vectorworks Design Series is installed, images can be added to worksheets to provide graphic views of the objects in a drawing. The images can be either thumbnail versions of objects, or samples of the 2D attributes of objects. For example, you might create a window schedule that shows top and front thumbnail views of each type of window in your drawing, in addition to data about each window.

Worksheets can have two types of rows: spreadsheet and database. The cells in a spreadsheet row contain constants (text or numbers), or formulas. Database rows consist of a header row and sub-rows, and they show data that are associated with specific drawing objects. Images can be inserted in either type of row.

To insert an image in a cell, use the image function, which works the same as other standard worksheet functions; see "Entering Formulas in Worksheet Cells" on page 1335.

Like other functions, you can enter an image function manually on the Formula bar, or use commands and dialog boxes to build a formula that uses the image function.

A cell that uses the image function in a formula can include either a text expression or an image, but not both.

To manually enter an image formula:

1. Select the cell.
2. Enter the image function (=IMAGE). The entry automatically displays in the worksheet Formula bar. If the cell is in a database header row, no further entry is required.
3. If the cell is in a spreadsheet row, enter the rest of the formula to specify which object to display. For example, to display an image of the symbol called K-02221, enter this:

   =IMAGE(S='K-02221')
4. When the formula is complete, click the green check mark to validate the entry. To cancel an entry, click the red X.
5. The formula executes, and the image(s) display (**Auto-recalc** must be selected in the worksheet preferences).
6. Customize the image display as described in "Entering Data in Spreadsheet Cells" on page 1333.

To enter a formula with the **Image Function** and **Criteria** commands:

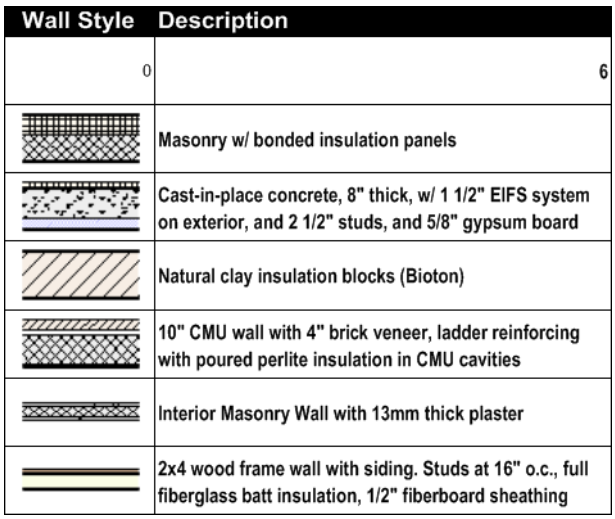
1. Select the cell.
2. Select **Insert > Image Function** from the **Worksheet** menu.

   The image function is placed in the worksheet Formula bar and the cursor is placed between the parentheses, awaiting an argument. If the cell is in a database header row, no further entry is required.
3. If the cell is in a spreadsheet row, enter criteria to specify which drawing object to display. Select **Insert > Criteria** from the **Worksheet** menu.
4. If an object is selected when the **Criteria** command is selected, the Paste Attributes dialog box opens. Otherwise, proceed to step 5.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Do one of the following:

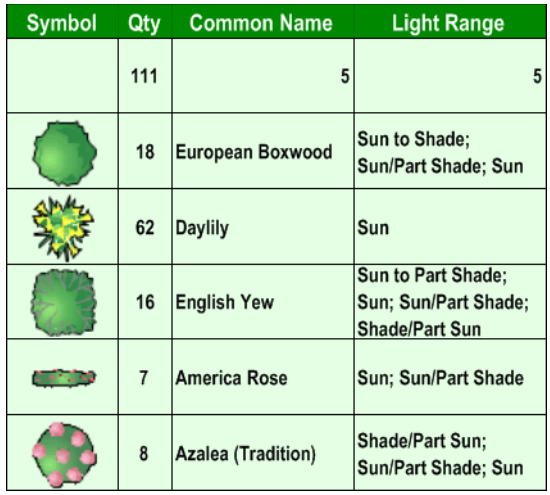
- To use attributes of the selected object as the only selection criteria, select the attributes and click **OK**. Proceed to step 5.
- To specify other criteria, or to use attributes of other objects in the drawing, click **Custom**.

5. The Criteria dialog box opens. For each set of criteria, select the choices that apply. Click **More Choices** to specify additional criteria sets. Click **Fewer Choices** to remove added criteria sets. Click **OK** to add the criteria to the function argument.
6. When the formula is complete, click the green check mark to validate the entry. To cancel the entry, click the red X.
7. The formula executes, and the image(s) display (**Auto-recalc** must be selected in the worksheet preferences).
8. Customize the image display as described in “Entering Data in Spreadsheet Cells” on page 1333.



Wall Style	Description
0	6
	Masonry w/ bonded insulation panels
	Cast-in-place concrete, 8" thick, w/ 1 1/2" EIFS system on exterior, and 2 1/2" studs, and 5/8" gypsum board
	Natural clay insulation blocks (Bioton)
	10" CMU wall with 4" brick veneer, ladder reinforcing with poured perlite insulation in CMU cavities
	Interior Masonry Wall with 13mm thick plaster
	2x4 wood frame wall with siding. Studs at 16" o.c., full fiberglass batt insulation, 1/2" fiberboard sheathing

Worksheet that includes 2D attribute images of wall styles



Symbol	Qty	Common Name	Light Range
	111	5	5
	18	European Boxwood	Sun to Shade; Sun/Part Shade; Sun
	62	Daylily	Sun
	16	English Yew	Sun to Part Shade; Sun; Sun/Part Shade; Shade/Part Sun
	7	America Rose	Sun; Sun/Part Shade
	8	Azalea (Tradition)	Shade/Part Sun; Sun/Part Shade; Sun

Worksheet that includes thumbnail images of plant symbols

Click here for a video tip about this topic (Internet access required).

Worksheet Functions

Worksheet functions take an argument, perform an action, and return a value or values. There are two basic types of functions: those that use the value(s) you enter, and those that use information from objects in the drawing. The arguments required by the two function types are different.

- **Number or text arguments:** Functions that begin with a lower case letter typically require a number value or a cell address as the argument. For example, the acos function returns the arccosine of the value that is specified in the function argument. The argument you enter can be a mathematical expression (such as 3/5), an address of a cell that contains a number (such as A12), or an actual number. The argument for all trigonometry functions must be in radians.
- **Criteria arguments:** Functions that begin with a capital letter must be applied to one or more specific objects in the drawing. In a cell in a database header row, a function is automatically applied to the object listed in each sub-row, so no criteria argument is required.

However, in a spreadsheet cell, you must enter criteria to select the objects the function applies to. For example, the Area function returns the combined area of all 2D objects that meet the criteria. To specify which objects to obtain the area of, either use the **Insert > Criteria** command on the **Worksheet** menu, or enter the criteria manually. For details about how to specify criteria such as the object type, class, or visibility, see the developer oriented documentation here:

http://developer.vectorworks.net/index.php/VS:Search_Criteria#Search_Criteria_Tables

http://developer.vectorworks.net/index.php/VS:Function_Reference_Appendix#attrCrit

The following table lists all of the worksheet functions available, as well as what kind of argument the function takes.

You may want to display an attribute associated with a drawing object in the worksheet (such as the object's class, or which layer it is on); see "Retrieving Object Attributes in a Worksheet" on page 1341.

Function (argument)	Description	Example	Related Functions
acos(number)	The arccosine of a number. The arccosine is the angle whose cosine is number. The returned angle is given in radians in the range 0 to pi. Number is the cosine of the angle, and must be from -1 to 1.	`=acos(3/5)` (returns the angle for which the cosine value is 3/5)	cos

Angle(criteria)	The angle (measured from horizontal) of the objects that meet the specified criteria, in degrees.
	Use this function to return the angles of lines and walls (measured from horizontal), the span angles of arcs, and the slope angles of slabs.
		Database header cell:
		`=Angle`
		(returns the angle of each object in the database)
		Spreadsheet cell:
		`=Angle((t=arc)&(n='arc-1'))`
		(returns the sweep angle of the arc object named "arc-1" in the drawing)	

Area(criteria)	The total area of 2D objects that meet the specified criteria, based on the Area units in the Units dialog box	**Database header cell:**
		`=Area`
		(returns the area of each object in the database)
		Spreadsheet cell:
		`=Area(t=rect)`
		(returns the combined area of all rectangle objects in the drawing)
			Perim

Function (argument)	Description	Example	Related Functions
asin(number)	The arcsine of a number. The arcsine is the angle whose sine is number. The returned angle is given in radians in the range -pi/2 to pi/2. To express the arcsine in degrees, use the rad2deg function (or multiply the result by 180/pi). Number is the sine of the angle and must be from -1 to 1.	`=asin(A3)` (returns the angle for which the sine value is given in cell A3)	sin
atan(number)	The arctangent of a number. The arctangent is the angle whose tangent is number. The returned angle is given in radians in the range -pi/2 to pi/2. To express the arctangent in degrees, use the rad2deg function (or multiply the result by 180/pi). Number is the tangent of the angle in question.	`=atan(4/3)` (returns the angle for which the tangent value is 4/3)	tan
average(number1, number2...)	The average (mean) of the arguments	`=average(85,70,95)` (returns the average of the three numbers)	max, min, sum

BotBound(criteria)	The bottom 2D boundary (minimum y coordinate) of the objects that meet the specified criteria	**Database header cell:**
		`=BotBound`
		(returns the bottom 2D boundary of each object in the database)
		Spreadsheet cell:
		`=BotBound(t=locus)`
		(returns the bottom 2D boundary of the locus that has the lowest bottom 2D boundary value in the drawing)
			LeftBound, RightBound, TopBound

Function (argument)	Description	Example	Related Functions

ComponentArea (criteria, index)	The area of one side of the specified wall or slab component, minus any holes.
	Index is the 1-based index identifying the component.
		Database header cell:
		`=ComponentArea(2)`
		(returns the area of the second component for each wall or slab object in the database)
		Spreadsheet cell:
		`=ComponentArea(t=wall,1)`
		(returns the combined area of the first components for all walls in the drawing)
			ComponentVolume, ComponentName

ComponentName (criteria, index)	The name of the specified wall or slab component.
	Index is the 1-based index identifying the component.
		Database header cell:
		`=ComponentName(2)`
		(returns the name of the second component for each wall or slab object in the database)
		Spreadsheet cell:
		`=ComponentName(t=wall,1)`
		(returns the name of the first component for all walls in the drawing)
			ComponentVolume, ComponentArea

ComponentVolume (criteria, index)	The volume of the specified component, minus any holes.
	Index is the 1-based index identifying the component.
		Database header cell:
		`=ComponentVolume(2)`
		(returns the volume of the second component for each wall or slab object in the database)
		Spreadsheet cell:
		`=ComponentVolume(t=wall,1)`
		(returns the combined volume of the first components for all walls in the drawing)
			ComponentArea, ComponentName

concat(text1, text2, text3)	Joins several text strings into one text string	`=concat(B3,', ',B4)` (returns the contents of cells B3 and B4 as a single string, separated by a comma and a space)	
cos(number)	The cosine of a given angle. Number is the angle in radians for which the cosine is calculated.	`=cos(deg2rad(23))` (converts a 23-degree angle to its radian equivalent, and returns the cosine of the angle)	acos

Function (argument)	Description	Example	Related Functions

Count(criteria)	The number of objects that meet the specified criteria	**Database header cell:**
		`=Count`
		(returns the total number of objects for each row in the database)
		Spreadsheet cell:
		`=Count(s='simple sofa')`
		(returns the total number of symbol objects named “simple sofa” in the drawing)	

CurtWallFrameLength (criteria, class)	The combined length of the curtain wall frames that meet the specified criteria and are in the specified class. To find all frames in a curtain wall, use an empty class name.	**Database header cell:**
		`=CurtWallFrameLength('')`
		(returns the combined length of the curtain wall frames for each curtain wall in the database)
		Spreadsheet cell:
		`=CurtWallFrameLength(t=wall, '')`
		(returns the combined length of the curtain wall frames for all curtain walls in the drawing)
			CurtWallPnlAreaNet, CurtWallPnlAreaGross

CurtWallPnlAreaGross (criteria, class)	The combined gross area of the curtain wall panels in the walls that meet the specified criteria and are in the specified class. The gross area includes portions of the panel covered by frames. To find all panels in a curtain wall, use an empty class name.	**Database header cell:**
		`=CurtWallPnlAreaGross('')`
		(returns the combined gross area of the curtain wall panels for each curtain wall in the database)
		Spreadsheet cell:
		`=CurtWallPnlAreaGross(t=wall, '')`
		(returns the combined gross area of the curtain wall panels for all curtain walls in the drawing)
			CurtWallFrameLength, CurtWallPnlAreaNet

Function (argument)	Description	Example	Related Functions

CurtWallPnlAreaNet (criteria, class)	The net area of the curtain wall panels in the walls that meet the specified criteria and are in the specified class. The net area includes only the visible area bounded by frames. To find all panels in a curtain wall, use an empty class name.	**Database header cell:**
		`=CurtWallPnlAreaNet ('Class-1')`
		(returns the combined net area of the curtain wall panels assigned to the class "Class-1" for each curtain wall in the database)
		Spreadsheet cell:
		`=CurtWallPnlAreaNet(t=wall, 'Class-1')`
		(returns the combined net area of the curtain wall panels assigned to the class "Class-1" for all curtain walls in the drawing)
			CurtWallFrameLength, CurtWallPnlAreaGross

deg2rad(number)	Converts a number from degrees to radians. Number is the value in degrees to be converted to radians.	`=deg2rad(47)` (converts the 47-degree angle measurement to its radian equivalent)	
exp(number)	e raised to the power of number. The constant e equals 2.71828182845904, the base of the natural logarithm. Number is the exponent applied to the base e.	`=exp(2)` (returns the numeric value of e raised to the power of 2)	ln
GetIfcProperty (criteria) (Vectorworks Architect or Landmark required)	The value of a specific IFC property associated with an IFC object. The criteria is a string with two elements separated by a period. The first element is either an IFC entity or PSet name, and the second element is the name of the IFC property.	`=GETIFCPROPERTY ('ifcfurnishingelement.name')` (returns the Name value for IFC objects whose IFC entity is IfcFurnishingElement)	

Function (argument)	Description	Example	Related Functions

Height(criteria)	The combined delta y (height) of objects that meet the specified criteria	**Database header cell:**
		`=Height`
		(returns the height (delta y) for each object in the database)
		Spreadsheet cell:
		`=Height(sel=true)`
		(returns the combined height (delta y) value of the selected objects in the drawing)
			Width

if ((logical_test), value_if_true, value_if_false)	Use value_if_true if logical_test is true, value_is_false if logical_test is false.
	Use this function to conduct conditional tests on values and formulas and to branch based on the results of that test. The outcome of the test determines the value returned by the If function. The logical_test can be any value or expression that can be evaluated to true or false. Up to seven If statements can be nested as value_if_true, value_if_false arguments. Boolean statements within an if statement must be in parentheses. Text within an if statement should be enclosed within quotation marks.
		`=if((C7>100),100,C7)`
		when commas are used as decimal separators by the operating system, use semicolons instead:
		`=if((C7>100);100;C7)`
		(if the value in cell C7 is greater than 100, the value in this cell is 100; otherwise, the value in this cell is the same as the value in cell C7)	

Image(criteria)
(Vectorworks Design Series required)
	The image associated with the object that meets the specified criteria. In the cell format, specify whether to show a thumbnail of the object, or the 2D attributes applied to the object.	**Database header cell:**
		`=Image`
		(returns the image for each object in the database)
		Spreadsheet cell:
		`=Image(s='cabinet')`
		(returns the image of the symbol named “Cabinet”)	

Function (argument)	Description	Example	Related Functions
int(number)	Removes any fractional part of a number. Number is the real number to be changed to an integer.	`=int(B9)` (returns the value in cell B9 without its fractional component)	round

IsFlipped(criteria)	The flipped state of the objects that meet the specified criteria	**Database header cell:**
		`=IsFlipped`
		(returns the flip state for each object in the database)
		Spreadsheet cell:
		`=IsFlipped(PON=window)`
		(returns the total number of window objects in the drawing that are flipped)	

LeftBound(criteria)	The left side 2D boundary (minimum x coordinate) of the objects that meet the specified criteria	**Database header cell:**
		`=LeftBound`
		(returns the left 2D boundary for each object in the database)
		Spreadsheet cell:
		`=LeftBound(t=locus)`
		(returns the left 2D boundary of the leftmost locus in the drawing)
			BotBound, TopBound, RightBound

Length(criteria)	The length of lines, walls, or path-based objects that meet the specified criteria	**Database header cell:**
		`=Length`
		(returns the length for each object in the database)
		Spreadsheet cell:
		`=Length(t=line)`
		(returns the total length of all line objects in the drawing)	

ln(number)	The natural logarithm (base e). Number is the positive real number for which the logarithm is calculated.	`=ln(12)` (returns the natural logarithm of 12)	exp
log(number)	The base 10 logarithm. Number is the positive real number for which the logarithm is calculated.	`=log(2)` (returns the base 10 logarithm of 2)	ln

Function (argument)	Description	Example	Related Functions
max(number1, number2,...)	The largest number in the list of arguments. Number is 1 – 14 numbers for which the maximum value is to be found.	`=max(C5,C7,C9)` (returns the largest of the numbers that are in cells C5, C7, and C9)	min
min(number1, number2,...)	The smallest number in the list of arguments. Number is 1 – 14 numbers for which the minimum value is to be found.	`=min(C5,C7,C9)` (returns the smallest of the numbers that are in cells C5, C7, and C9)	max

ObjectType(criteria)	The numeric object type ID of objects that meet the specified criteria
	For a list of object type IDs, see the developer oriented documentation here:
	http://developer.vectorworks.net/index.php/VS:Function_Reference_Appendix#objects
		Database header cell:
		`=ObjectType`
		(returns the object type value for each object in the database)
		Spreadsheet cell:
		`=ObjectType(sel=true)`
		(returns the object type value of the selected object; for example, the object type value for a light is 81)	

Perim(criteria)	The combined perimeter of objects that meet the specified criteria	**Database header cell:**
		`=Perim`
		(returns the perimeter for each object in the database)
		Spreadsheet cell:
		`=Perim(sel=true)`
		(returns the total perimeter of all selected objects)	

rad2deg(number)	Converts a number from radians to degrees. Number is the value in radians to be converted to degrees.	`=rad2deg(0.5235987)` (converts the radian angle measurement to its degree equivalent)	

Function (argument)	Description	Example	Related Functions

RightBound(criteria)	The right side 2D boundary (maximum x coordinate) of the objects that meet the specified criteria	**Database header cell:**
		`=RightBound`
		(returns the right 2D boundary for each object in the database)
		Spreadsheet cell:
		`=RightBound(t=rect)`
		(returns the right 2D boundary of the rightmost rectangle in the drawing)
			BotBound, TopBound, LeftBound

RoofArea_Heated (criteria)	The heated area of the roof (minus the eve overhang) along the slope, combined for all objects that meet the specified criteria	**Database header cell:**
		`=RoofArea_Heated`
		(returns the heated area for each roof and roof face object in the database)
		Spreadsheet cell:
		`=RoofArea_Heated (st=roofface)`
		(returns the combined heated area of all roof face objects in the drawing)
			RoofArea_HeatedProj

RoofArea_HeatedProj (criteria)	The heated area of the roof (minus the eve overhang) projected to the layer plane, combined for all objects that meet the specified criteria	**Database header cell:**
		`=RoofArea_HeatedProj`
		(returns the heated area for each roof and roof face object in the database, as projected to the layer plane)
		Spreadsheet cell:
		`=RoofArea_Heatedproj(t=roof)`
		(returns the combined heated area of all roof objects in the drawing, as projected to the layer plane)
			RoofArea_Heated

RoofArea_Total (criteria)	The total area of the roof along the slope	**Database header cell:**
		`=RoofArea_Total`
		(returns the total area for each roof and roof face object in the database)
		Spreadsheet cell:
		`=RoofArea_Total(st=roofface)`
		(returns the combined total area of all roof face objects in the drawing)
			RoofArea_TotalProj

Function (argument)	Description	Example	Related Functions

RoofArea_TotalProj (criteria)	The total area of the roof, projected to the layer plane	**Database header cell:**
		`=RoofArea_TotalProj`
		(returns the total area for each roof and roof face object in the database, as projected to the layer plane)
		Spreadsheet cell:
		`=RoofArea_Totalproj(t=roof)`
		(returns the combined total area of all roof objects in the drawing, as projected to the layer plane)
			RoofArea_Total

round(number)	Rounds the specified number to the nearest whole number	`=round(D11)` (returns the value in cell D11 rounded to the nearest whole number)	int
sin(number)	The sine of a given angle. Number is the angle in radians for which the sine is calculated.	`=sin(deg2rad(32))` (converts a 32-degree angle to its radian equivalent, and returns the sine of the angle)	asin
SlabStyleName (Vectorworks Architect required)	The name of a slab style	**Database header cell:** `=SlabStyleName` (returns the name of the slab style for each slab object in the database)	

SlabThickness (criteria)
(Vectorworks Architect required)
	The combined thickness of slab objects (floors and roof faces) that meet the specified criteria	**Database header cell:**
		`=SlabThickness`
		(returns the thickness for each object in the database)
		Spreadsheet cell:
		`=SlabThickness(PON=slab)`
		(returns the combined thickness of all slab objects in the drawing)	

sqrt(number)	A positive square root. Number is the number for which the square root is calculated.	`=sqrt(D27)` (returns the square root of the number in cell D27)	
Substring(text/ function, delimiter, index)	Splits a single string into an array of strings using a delimiter, and outputs each string at the specified index	`=SUBSTRING(('kitchen;bedroom; bathroom;basement', ';', 2)` (returns “bedroom,” which is the second substring in the specified string)	

Function (argument)	Description	Example	Related Functions
sum(number1, number2,...)	The sum of all numbers in the list of arguments. Number is 1 – 14 numbers for which the sum is calculated.	`=sum(A2,A10..A12)` (returns the sum of the numbers contained in cells A2, A10, A11, and A12)	Average

SurfaceArea(criteria)	The total surface area of all objects that meet the criteria, based on the Area units in the Units dialog box	**Database header cell:**
		`=SurfaceArea`
		(returns the surface area for each object in the database)
		Spreadsheet cell:
		`=SurfaceArea(st=sphere)`
		(returns the total surface area of all sphere objects in the drawing)	

tan(number)	The tangent of the given angle. Number is the angle in radians for which the tangent is calculated.	`=tan(deg2rad(32))` (converts a 32-degree angle to its radian equivalent, and returns the tangent of the angle)	atan

TopBound(criteria)	The top 2D boundary (maximum y coordinate) of the objects that meet the specified criteria	**Database header cell:**
		`=TopBound`
		(returns the top 2D boundary for each object in the database)
		Spreadsheet cell:
		`=TopBound(sel=true)`
		(returns the top 2D boundary of the topmost selected object)
			BotBound, LeftBound, RightBound

value(text)	Converts a text string that represents a number to a number	`=value('2e3')` (returns the numeric value of 2 times 10 raised to the power of 3)	

Volume(criteria)	The total volume of all objects that meet the criteria, based on the Volume units in the Units dialog box.	**Database header cell:**
		`=Volume`
		(returns the volume for each object in the database)
		Spreadsheet cell:
		`=Volume(t=xtrd)`
		(returns the total volume of all extrude objects in the drawing)	

Function (argument)	Description	Example	Related Functions

WallArea_Gross (criteria)	The combined 2D gross surface area of one wall face for walls that meet the specified criteria	**Database header cell:**
		`=WallArea_Gross`
		(returns the gross surface area of one wall face for each wall object in the database)
		Spreadsheet cell:
		`=WallArea_Gross(t=wall)`
		(returns the combined gross surface area of one wall face for all walls in the drawing)
			WallArea_Net

WallArea_Net (criteria)	The combined 2D net surface area of one wall face, without door, window, and inserted symbol areas, for walls that meet the specified criteria	**Database header cell:**
		`=WallArea_Net`
		(returns the net surface area of one wall face, minus inserted object areas, for each object in the database)
		Spreadsheet cell:
		`=WallArea_Net(t=wall)`
		(returns the combined net surface area of one wall face, minus inserted object areas, for all walls in the drawing)
			WallArea_Gross

WallAverageHeight (criteria)	The average height of a wall, including wall peaks and different starting and ending heights	**Database header cell:**
		`=WallAverageHeight`
		(returns the average height for each wall object in the database)
		Spreadsheet cell:
		`=WallAverageHeight((t=wall)&(sel=true))`
		(returns the average height of all walls that are selected in the drawing)	

WallStyleName (criteria) (Vectorworks Architect or Landmark required)	The name of a wall style	**Database header cell:** `=WallStyleName` (returns the name of the wall style for each wall object in the database)	

Function (argument)	Description	Example	Related Functions

WallThickness (criteria)	The combined thickness of wall objects that meet the specified criteria	**Database header cell:**
		`=WallThickness`
		(returns the thickness for each wall object in the database)
		Spreadsheet cell:
		`=WallThickness(t=wall)`
		(returns the combined thickness of all walls in the drawing)	

Width(criteria)	The combined delta x (width) of objects that meet the specified criteria	**Database header cell:**
		`=Width`
		(returns the width (delta x) for each object in the database)
		Spreadsheet cell:
		`=Width(sel=true)`
		(returns the combined width (delta x value) of the selected object)
			Height

XCenter(criteria)	The x coordinate of the center point of an object that meets the specified criteria	**Database header cell:**
		`=XCenter`
		(returns the x coordinate value of the center of the 2D boundary for each object in the database)
		Spreadsheet cell:
		`=XCenter(sel=true)`
		(returns the x coordinate value of the center of the 2D boundary of the selected object)
			YCenter, ZCenter, XCoordinate

XCoordinate(critiera)	The x coordinate of the insertion point for symbols, point plug-in objects, and loci. The returned value is relative to the user origin.	**Database header cell:** `=XCoordinate` (returns the x coordinate value for each object in the database)	YCoordinate, ZCoordinate, XCenter

Function (argument)	Description	Example	Related Functions

YCenter(criteria)	The y coordinate of the center point of an object that meets the specified criteria	**Database header cell:**
		`=YCenter`
		(returns the y coordinate value of the center of the 2D boundary for each object in the database)
		Spreadsheet cell:
		`=YCenter(sel=true)`
		(returns the y coordinate value of the center of the 2D boundary of the selected object)
			XCenter, ZCenter, YCoordinate

YCoordinate(criteria)	The y coordinate of the insertion point for symbols, point plug-in objects, and loci. The returned value is relative to the user origin.	**Database header cell:** `=YCoordinate` (returns the y coordinate value for each object in the database)	XCoordinate, ZCoordinate, YCenter

ZCenter(criteria)	The z coordinate of the center point of an object that meets the specified criteria	**Database header cell:**
		`=ZCenter`
		(returns the z coordinate value of the center of the 3D boundary for each object in the database)
		Spreadsheet cell:
		`=ZCenter(sel=true)`
		(returns the z coordinate value of the center of the 3D boundary of the selected object)
			XCenter, YCenter, ZCoordinate

ZCoordinate(criteria)	The z coordinate of the insertion point for symbols, point plug-in objects, and loci. The returned value is relative to the user origin.	**Database header cell:** `=ZCoordinate` (returns the z coordinate value for each object in the database)	XCoordinate, YCoordinate, ZCenter

Entering Data in Spreadsheet Cells
Entering Data in Database Rows

Importing Worksheets

Data from a wide variety of worksheet formats can be imported into a file, including worksheet data from other programs and worksheets from other Vectorworks files.

Importing Worksheet Data from Another Program

To import a worksheet into the current file, a blank worksheet must exist in the active drawing as a destination for the imported worksheet. During import, the spreadsheet data is imported, but the formulas (logic) are not.

To import worksheet data into a Vectorworks file:

1. From the Resource Browser, select **Resources > New Resource > Worksheet**.

   The Create Worksheet dialog box opens.
2. Specify the same number of rows and columns that are present in the worksheet to be imported.
3. Select **File > Import > Import Worksheet**.
4. Select the worksheet file to be imported. The worksheet must be in one of the following formats:
   - Tab delimited text (.txt)
   - Comma delimited text (.csv)
   - DIF (.dif)
   - SYLK (.slk)
5. Click **Open**. The worksheet is imported into the open worksheet in the Vectorworks file.

Importing Worksheets from Another Vectorworks File

To import a Vectorworks worksheet from another Vectorworks file:

1. From the Resource Browser, locate the file containing the worksheet using the **Resources > Find** command, or from a Favorites file. See “Accessing Existing Resources” on page 229 for more information.
2. Select the worksheet to import, and click **Import**. (Alternatively, drag the worksheet resource into the desired Vectorworks file to insert it as a drawing object; see “Worksheets as Graphic Objects” on page 1360.)

   The selected worksheet is imported into the current file.

Importing a Table from Microsoft Word

The following procedure is an example of how to import a table from Microsoft Word into a Vectorworks worksheet.

To import a Word table into a worksheet:

1. In Word, select the entire table to be imported.
2. In versions of Word prior to 2007, select **Table > Convert > Table to Text**. In Word 2007 and later, select the **Table Tools > Layout > Convert to Text**.

   The Convert Table to Text dialog box opens.
3. In the dialog box, click **Tabs** for the separation identifier.
4. Click **OK**.
5. Save the table.

   If the file consists of only the table, select **File > Save As**. In the Save As dialog box that opens, select **Text Only** from the **Save as type** list, and click **OK**.

   If the table is part of a larger file, copy the table to the Clipboard. Select **File > New**, and paste the table text into the new document. Save the file as a text file.
6. Return to the Vectorworks file and import the table as described in “Importing Worksheet Data from Another Program” on page 1359.

Exporting Worksheets

Vectorworks worksheet files can be exported and read by spreadsheet programs, such as Excel, as well as by some word processing programs. Because many of these programs have different format requirements, several export formats are available. These include comma delimited, tab delimited, merge, DIF, and SYLK. Using the tab delimited format, for example, creates a file that can be opened as a table in Microsoft Word.

To export a worksheet from a Vectorworks file:

1. Open the worksheet to export.
2. Select **File > Export > Export Worksheet**.
3. The Export Worksheet dialog box opens.

   Select the format for the export, and whether to export all rows or only the selected rows. Click **OK** and specify the name and location of the exported worksheet.
4. Click **Save**. The specified worksheet rows are exported to the desired location in the set format.

Worksheets as Graphic Objects

A worksheet can be included in a drawing as a graphic object. The worksheet object can be moved to any location on the drawing, but it cannot be resized. Use the Attributes palette to modify the fill, pen, and line thickness attributes of the entire worksheet object. To edit the contents of the worksheet, open it from the Resource Browser, or from the drawing itself.

To include a worksheet as an object on a drawing:

1. From the Resource Browser, select the desired worksheet.

   The worksheet must be a part of the current drawing. If the worksheet is part of another drawing, first import it into the current drawing. Select the worksheet from the Resource Browser and then select **Resources > Import**. (See “Importing Worksheet Data from Another Program” on page 1359.)
2. Select **Resources > Worksheet On Drawing**.

   The worksheet is now included in the drawing as a graphic object. It can be moved like any other graphic object and placed where desired. If the worksheet is currently open, the contents of the worksheet in the drawing are replaced with an X.

   Alternatively, double-click the worksheet in the Resource Browser to place it in the drawing file, or select the worksheet in the Resource Browser and drag it to the desired location in the drawing file. Both methods automatically select **Worksheet on Drawing** and place the worksheet as a graphic object. If you use the drag and drop method, the top left corner of the worksheet is placed at the point where the mouse is released in the drawing file.

To edit a worksheet included as a drawing object:

1. From the Resource Browser, select the desired worksheet.
2. Select **Resources > Open**.

   Alternatively, double-click the worksheet object in the drawing.
3. The worksheet is opened and available for editing. The contents of the worksheet in the drawing are replaced with an X. Close the worksheet to display the worksheet contents in the drawing.

Worksheet Tutorial: Creating a Wall Schedule

This simple tutorial explains how to create a customized schedule that links wall objects in a drawing to a database worksheet. The schedule will display data for wall areas, wall thicknesses, labor cost, material cost, taxes, and total cost per wall style.

The WorksheetTutorial.vwx file contains the data required to perform the tutorial steps. Though wall styles are a Vectorworks Design Series feature, any Vectorworks license can use the wall styles that already exist in the tutorial file. Download the file here (Internet connection required) and open it in Vectorworks to begin.

Step 1: Create a Blank Worksheet

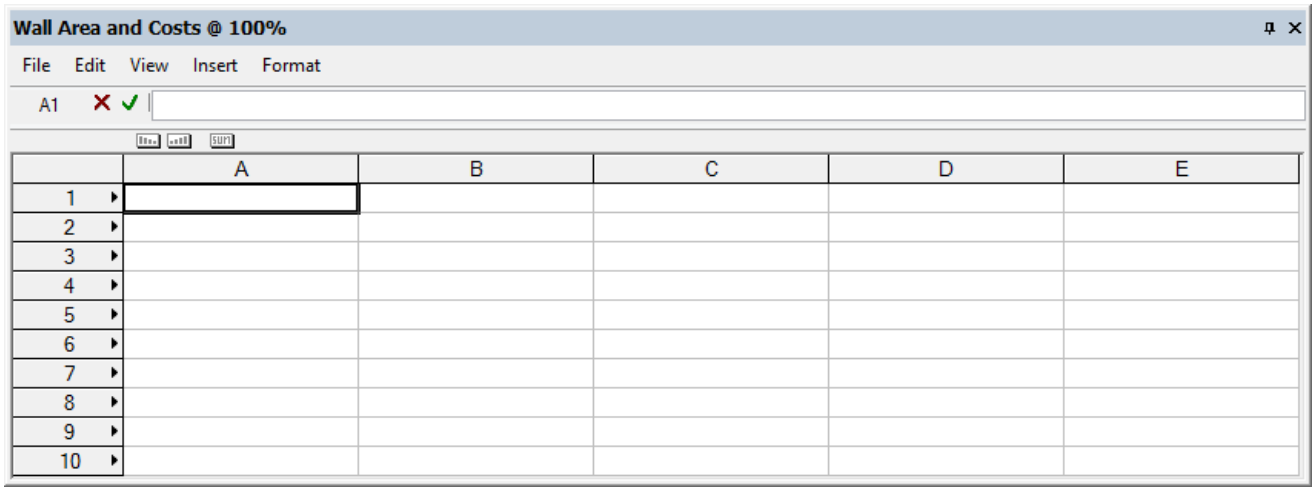
There are three ways to create a schedule using worksheets. This tutorial uses the first option below.

- Start with a blank worksheet, and create the schedule from scratch. See “Creating a Blank Worksheet” on page 1317.
- Create a schedule based on a common record format of a set of objects (wall data, in this example). This option allows you to select the criteria you wish to display from all the available object criteria. See “Creating Reports” on page 1316.
- Start with a preformatted schedule and customize it to achieve your goals. See “Creating Schedules” on page 1864.

To create a blank worksheet:

1. From the Resources menu in the Resource Browser, select **New Resource > Worksheet**. The Create Worksheet dialog box opens.
2. Enter “Wall Area and Costs” as the name for the new worksheet, and click **OK**. You will add more rows and columns later.

   A blank worksheet window opens.



Step 2: Set the Database Criteria

Next, create a database of the objects in the drawing from which to extract the wall area data. You can combine multiple criteria to collect the desired subset of objects.

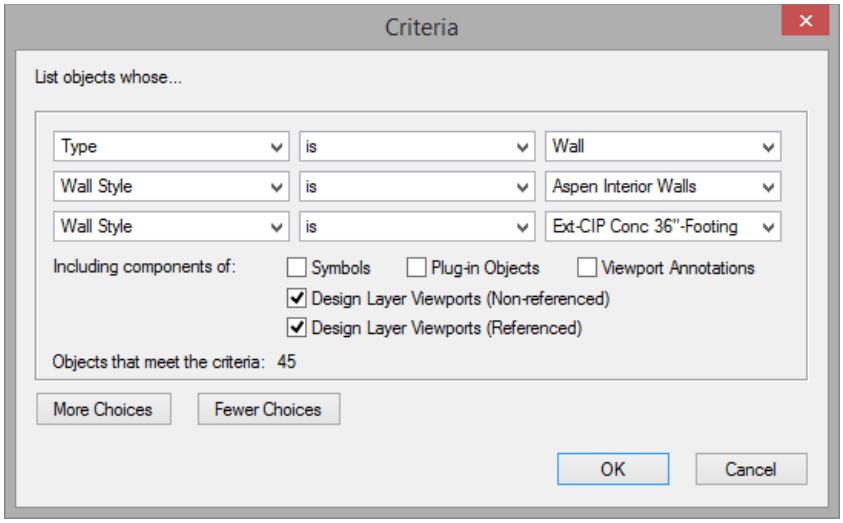
For this tutorial, a single database of wall objects will be created and limited to a specified set of wall styles.

> An alternative would be to create one database per wall style and include multiple databases in the same worksheet. However, for very large databases, it is recommended to create separate worksheets rather than include multiple databases into a single worksheet.

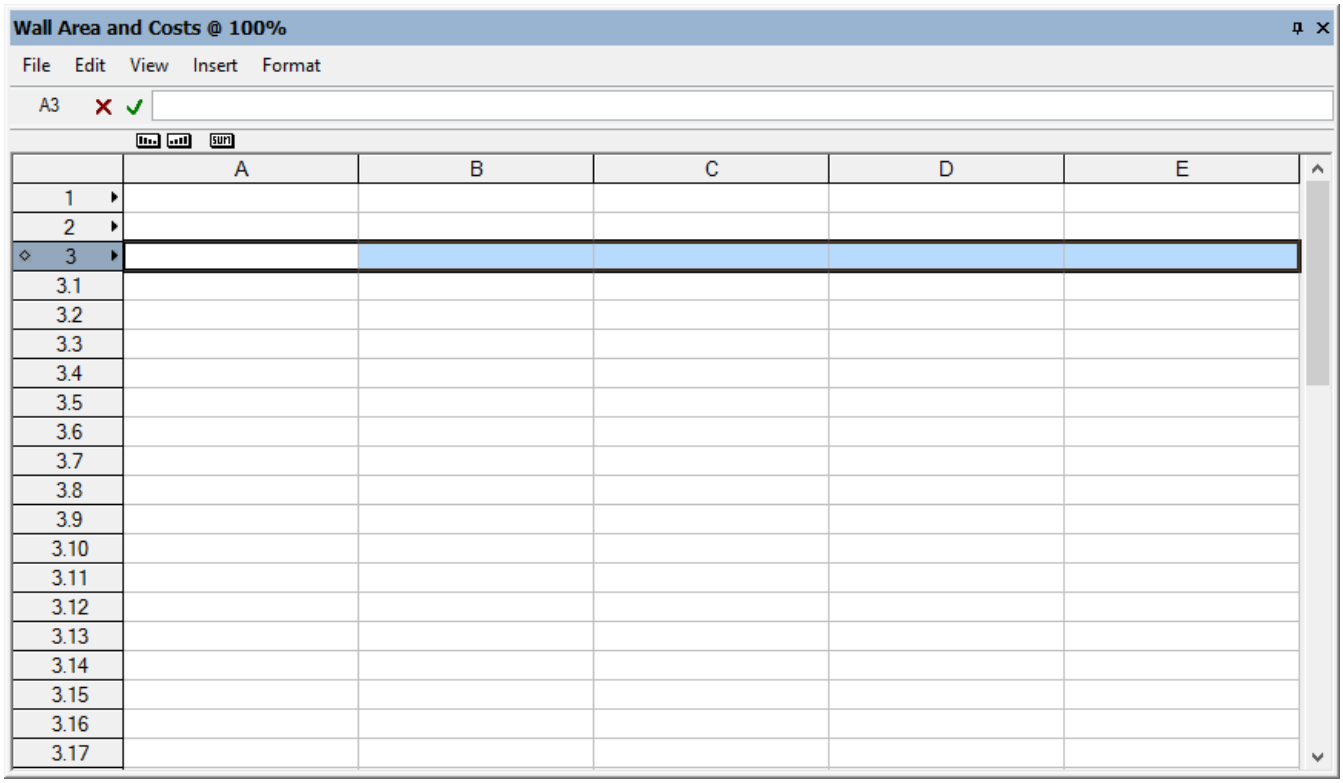
To set the database criteria:

1. Right-click (Windows) or Ctrl-click (Mac) on the header box for row 3.
2. From the **Row** context menu, select **Database**. The Criteria dialog box opens.
3. Set the three criteria options as follows:
   - Type
   - is
   - Wall
4. Click **More Choices**, and set the next three criteria as follows:
   - Wall Style
   - is
   - Aspen Interior Walls
5. Click **More Choices**, and set the next three criteria as follows:
   - Wall Style
   - is
   - Ext-CIP Conc 36”-Footing

   To include all wall styles in the schedule, do not enter criteria for walls and wall styles; instead, use the following criteria: Record, Wall Data, is present.



6. Click **OK** to set the criteria. The database of walls for the specified set of wall styles is created. The database header (row 3) now has a diamond next to the row number. Beneath row 3 are sub-rows for each object in the database (3.1 through 3.45).



Step 3: Expand the Worksheet

For this tutorial, you need to expand the worksheet. Since no data has been assigned to the columns yet, it does not matter where the columns are added.

Use one of the following methods to add three columns to the worksheet, for a total of eight.

- Select **Insert > Columns**. An empty column is added to the left of the current column.
- Right-click (Windows) or Ctrl-click (Mac) the column header where you want to add a column, and select **Insert Columns** from the context menu.
- Position the cursor at the bottom right corner of the worksheet to activate a special resize cursor; drag as needed to add columns to the right side of the worksheet.

Step 4: Use Worksheet Functions to Extract Data

Next, add database functions to the worksheet to extract the desired data from the database. Enter formulas for each column in the database header row cells. The database header row can be hidden before the worksheet is placed on the drawing.

For this tutorial, the following data will be extracted:

- Wall Style Name
- Gross Wall Area
- Net Wall Area
- Wall Thickness

To extract the data associated with the walls in the database:

1. Click the following cells and enter the formula shown to extract data for each item in the database. Be sure to include the equal sign (=) before each item.
   - In A3 enter `=WALLSTYLENAME`

- In B3 enter `=WALLAREA_GROSS`
- In C3 enter `=WALLAREA_NET`
- In D3 enter `=WALLTHICKNESS`

Alternatively, use the worksheet menu command **Insert > Function** to insert functions.

2. By default, numerical data is unformatted and must be formatted to display appropriate units. Formatting applied to database header row cells is automatically applied to all sub-rows for that column.

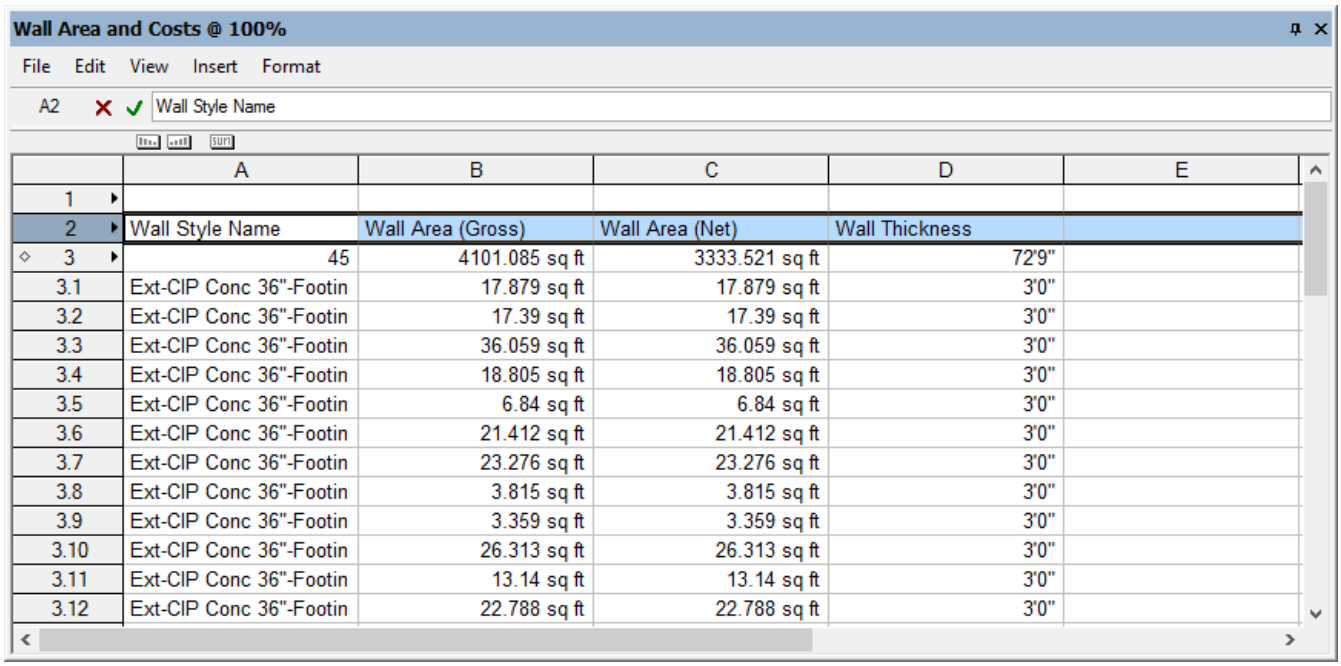
   Right-click (Windows) or Ctrl-click (Mac) each of the following cells, and select **Format Cells** from the context menu. The Format Cells dialog box opens. On the Number tab, select the format option shown and click **OK**.

   - For B3 select **Dimension Area**
   - For C3 select **Dimension Area**
   - For D3 select **Dimension**

   Alternatively, use the worksheet menu command **Format > Cells** to format cells.

3. Add labels for columns A through D by typing names in the cells in row 2.

   - In A2 enter `Wall Style Name`
   - In B2 enter `Wall Area (Gross)`
   - In C2 enter `Wall Area (Net)`
   - In D2 enter `Wall Thickness`



	A	B	C	D	E
1					
2	Wall Style Name	Wall Area (Gross)	Wall Area (Net)	Wall Thickness	
3	45	4101.085 sq ft	3333.521 sq ft	72'9"	
3.1	Ext-CIP Conc 36"-Footin	17.879 sq ft	17.879 sq ft	3'0"	
3.2	Ext-CIP Conc 36"-Footin	17.39 sq ft	17.39 sq ft	3'0"	
3.3	Ext-CIP Conc 36"-Footin	36.059 sq ft	36.059 sq ft	3'0"	
3.4	Ext-CIP Conc 36"-Footin	18.805 sq ft	18.805 sq ft	3'0"	
3.5	Ext-CIP Conc 36"-Footin	6.84 sq ft	6.84 sq ft	3'0"	
3.6	Ext-CIP Conc 36"-Footin	21.412 sq ft	21.412 sq ft	3'0"	
3.7	Ext-CIP Conc 36"-Footin	23.276 sq ft	23.276 sq ft	3'0"	
3.8	Ext-CIP Conc 36"-Footin	3.815 sq ft	3.815 sq ft	3'0"	
3.9	Ext-CIP Conc 36"-Footin	3.359 sq ft	3.359 sq ft	3'0"	
3.10	Ext-CIP Conc 36"-Footin	26.313 sq ft	26.313 sq ft	3'0"	
3.11	Ext-CIP Conc 36"-Footin	13.14 sq ft	13.14 sq ft	3'0"	
3.12	Ext-CIP Conc 36"-Footin	22.788 sq ft	22.788 sq ft	3'0"	

Various types of data can be extracted from Vectorworks objects into a worksheet database, as described in the following topics.

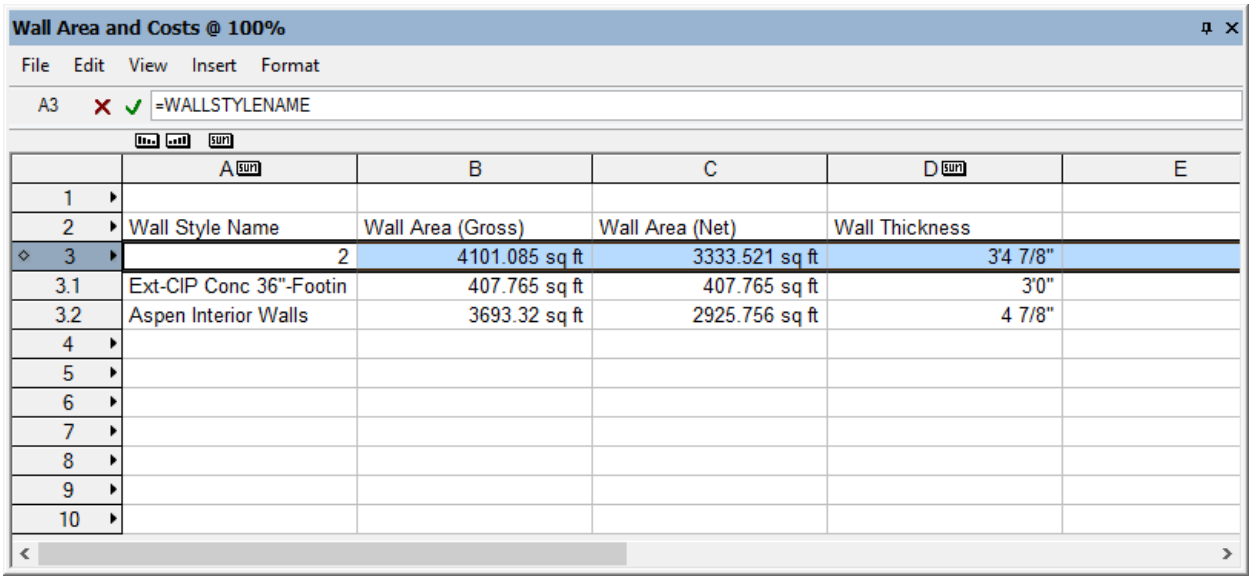
- “Worksheet Functions” on page 1344
- “Retrieving Object Attributes in a Worksheet” on page 1341
- “Retrieving Record Information in a Worksheet” on page 1342
- “Entering Formulas in Worksheet Cells” on page 1335

Step 5: Summarize Wall Styles

Instead of listing each wall individually in the database, you can summarize all walls with the same wall style, automatically calculating the total quantities for each, and shortening the list.

To summarize the wall styles:

1. Click on the header box for row 3 to select it. Three icons display just below the Formula bar.
2. Click and drag the Summarize icon to the header box for column A, which contains wall styles.
3. The number of sub-rows is reduced to only two (one row per wall style). The numerical values in columns B, C, and D are now sums. While this is desired for the wall area gross and net columns, you probably want to show the thickness value as the thickness of an individual wall, rather than all walls combined. To do so, apply another Summarize icon to column D.



	A	B	C	D	E
1					
2	Wall Style Name	Wall Area (Gross)	Wall Area (Net)	Wall Thickness	
3	2	4101.085 sq ft	3333.521 sq ft	3'4 7/8"	
3.1	Ext-CIP Conc 36"-Footin	407.765 sq ft	407.765 sq ft	3'0"	
3.2	Aspen Interior Walls	3693.32 sq ft	2925.756 sq ft	4 7/8"	
4					
5					
6					
7					
8					
9					
10					

Step 6: Use Formulas to Calculate Costs

Next, calculate costs using worksheet operations and formulas.

For this tutorial, the following data will be calculated:

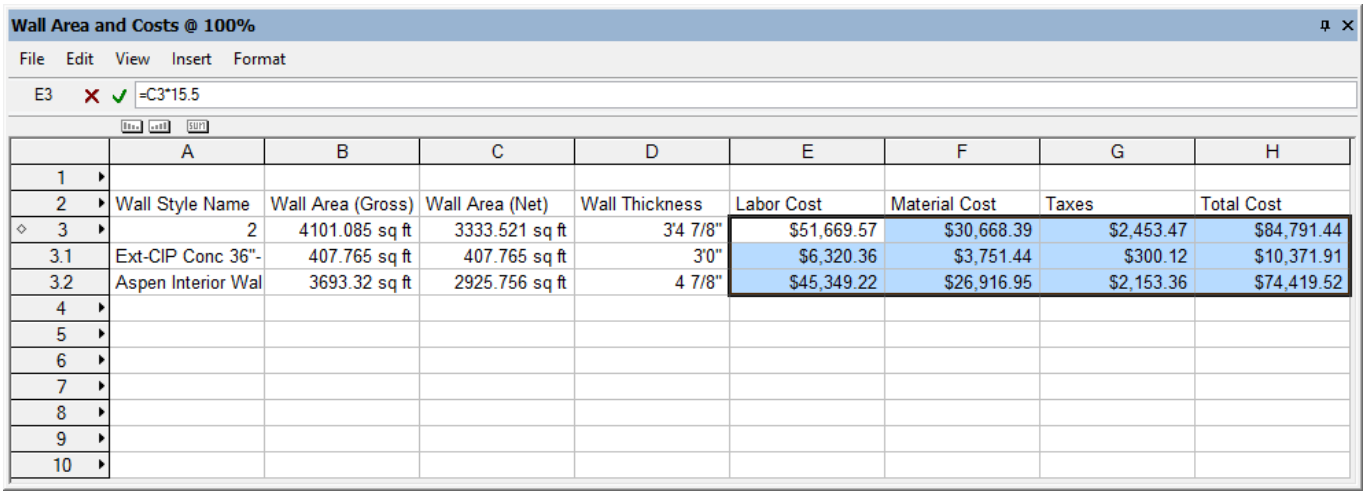
- Labor cost per wall style
- Material cost per wall style
- Taxes
- Total cost

To calculate costs with formulas:

1. Add labels for columns E through H by typing names in the cells in row 2.
   - In E2 enter `Labor Cost`
   - In F2 enter `Material Cost`
   - In G2 enter `Taxes`
   - In H2 enter `Total Cost`
2. Click the following cells and enter the formula shown to determine the cost sums. Be sure to include the equal sign (=) before each item.
   - In E3 enter `=C3*15.5` (multiply the value in C3 by 15.5, the estimated rate of labor cost per area unit)
   - In F3 enter `=C3*9.2` (multiply the value in C3 by 9.2, the estimated material cost per area unit)
   - In G3 enter `=F3*.08` (multiply the value in F3 by 0.08, the estimated tax rate)
   - In H3 enter `=E3+F3+G3` (calculate the total cost of labor, material, and taxes)

   Instead of entering the tax rate directly into the calculation, an alternative and more flexible approach is to set the tax rate in a separate spreadsheet cell and simply reference it in the calculation.

3. Format all of the cost data the same way. Select the four cost header cells (E3 through H3), and then right-click (Windows) or Ctrl-click (Mac) and select **Format Cells** from the context menu. The Format Cells dialog box opens. On the Number tab, select the options shown and click **OK**.
   - Select **Decimal**
   - In **Dec. Places**, enter 2
   - Select **Use Commas**
   - In **Leader**, enter $ (dollar sign)



	A	B	C	D	E	F	G	H
1								
2	Wall Style Name	Wall Area (Gross)	Wall Area (Net)	Wall Thickness	Labor Cost	Material Cost	Taxes	Total Cost
3	2	4101.085 sq ft	3333.521 sq ft	3'4 7/8"	$51,669.57	$30,668.39	$2,453.47	$84,791.44
3.1	Ext-CIP Conc 36"-	407.765 sq ft	407.765 sq ft	3'0"	$6,320.36	$3,751.44	$300.12	$10,371.91
3.2	Aspen Interior Wal	3693.32 sq ft	2925.756 sq ft	4 7/8"	$45,349.22	$26,916.95	$2,153.36	$74,419.52
4								
5								
6								
7								
8								
9								
10								

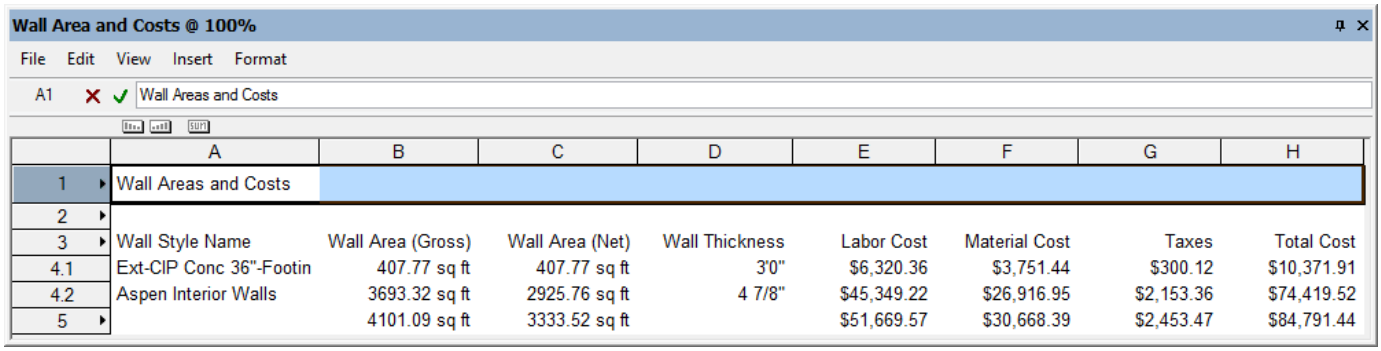
Step 7: Set up Column Totals, and Add and Delete Rows

Next, set up totals at the bottom of the columns as appropriate. The cells in the database header row (in this case, row 3) display sums for all of the database columns. Reference these database header row cells in spreadsheet cells to set up the totals.

To set up the totals:

1. Click the following cells and enter the formula shown to show sums from the database header row. Be sure to include the equal sign (=) before each item.
   - In B4 enter `=B3`
   - In C4 enter `=C3`
   - In E4 enter `=E3`
   - In F4 enter `=F3`
   - In G4 enter `=G3`
   - In H4 enter `=H3`
2. Select cells B4 and C4, right-click (Windows) or Ctrl-click (Mac), and select **Format Cells** from the context menu. On the Number tab, select **Dimension Area** and click **OK**.
3. Select cells E4 through H4, right-click (Windows) or Ctrl-click (Mac), and select **Format Cells** from the context menu. On the Number tab, select the options shown and click **OK**.
   - Select **Decimal**
   - In **Dec. Places**, enter 2
   - Select **Use Commas**
   - In **Leader**, enter $ (dollar sign)
4. Click cell A1 and enter `Wall Areas and Costs` as the schedule title.
5. Select row 2, right-click (Windows) or Ctrl-click (Mac), and select **Insert Rows** from the context menu to add an empty row between the schedule title and the column labels.

6. Select the empty rows at the bottom of the worksheet (6 through 11), right-click (Windows) or Ctrl-click (Mac), and select **Delete Rows** from the context menu.
7. Select **View > Database Headers** and then **View > Grid Lines** to hide the database header row and grid lines in the table.

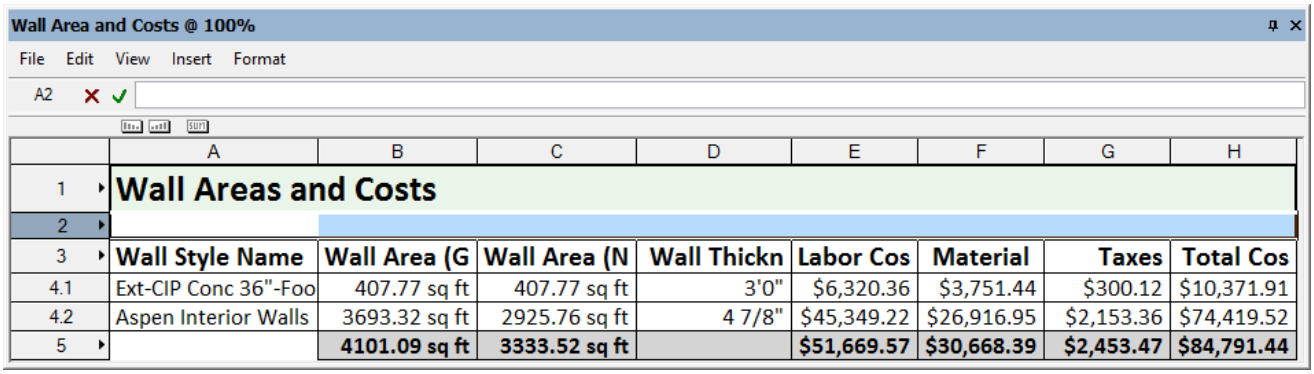


	A	B	C	D	E	F	G	H
1	Wall Areas and Costs							
2								
3	Wall Style Name	Wall Area (Gross)	Wall Area (Net)	Wall Thickness	Labor Cost	Material Cost	Taxes	Total Cost
4.1	Ext-CIP Conc 36"-Footin	407.77 sq ft	407.77 sq ft	3'0"	$6,320.36	$3,751.44	$300.12	$10,371.91
4.2	Aspen Interior Walls	3693.32 sq ft	2925.76 sq ft	4 7/8"	$45,349.22	$26,916.95	$2,153.36	$74,419.52
5		4101.09 sq ft	3333.52 sq ft		$51,669.57	$30,668.39	$2,453.47	$84,791.44

Step 8: Format the Worksheet

Finally, select cells and use the **Format Cells** command to format the worksheet as needed. You can change the font, font style, size, and color to format the text. Add cell borders, and change cell background color, as desired. Change the text alignment in cells, and resize rows and columns, if necessary.

See “Formatting Worksheet Cells” on page 1330 for details.



	A	B	C	D	E	F	G	H
1	**Wall Areas and Costs**							
2								
3	**Wall Style Name**	**Wall Area (G**	**Wall Area (N**	**Wall Thickn**	**Labor Cos**	**Material**	**Taxes**	**Total Cos**
4.1	Ext-CIP Conc 36"-Foo	407.77 sq ft	407.77 sq ft	3'0"	$6,320.36	$3,751.44	$300.12	$10,371.91
4.2	Aspen Interior Walls	3693.32 sq ft	2925.76 sq ft	4 7/8"	$45,349.22	$26,916.95	$2,153.36	$74,419.52
5		**4101.09 sq ft**	**3333.52 sq ft**		**$51,669.57**	**$30,668.39**	**$2,453.47**	**$84,791.44**

Wall Areas and Costs

Wall Style Name	Wall Area (Gross)	Wall Area (Net)	Wall Thickness	Labor Cost	Material Cost	Taxes	Total Cost
Ext-CIP Conc 36"-Footing	407.77 sq ft	407.77 sq ft	3'0"	$6,320.36	$3,751.44	$300.12	$10,371.91
Aspen Interior Walls	3693.32 sq ft	2925.76 sq ft	4 7/8"	$45,349.22	$26,916.95	$2,153.36	$74,419.52
	4101.09 sq ft	**3333.52 sq ft**		**$51,669.57**	**$30,668.39**	**$2,453.47**	**$84,791.44**

Formatted worksheet placed in drawing

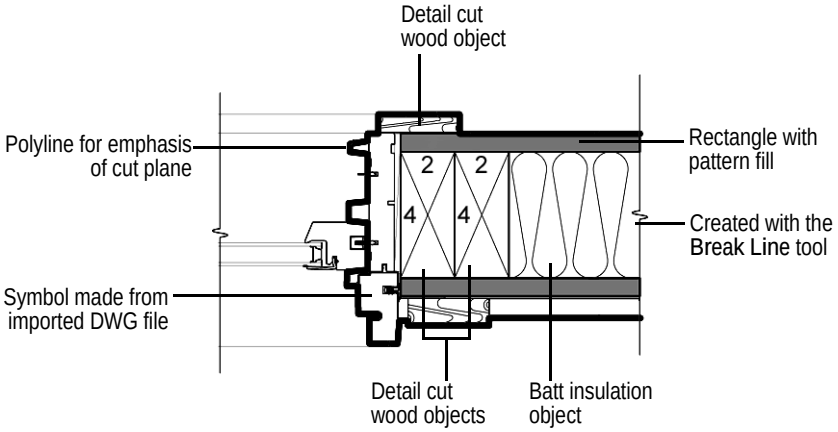
Detail Drawings

To create detail drawings for architectural projects, first use the **Create Standard Viewports** command (Vectorworks Architect required) to automatically create the necessary detail sheet layers. For a complete description of viewports, see “Creating Sheet Layer Viewports” on page 1616.

When using viewports:

- The **Create Standard Viewports** command can automatically create the necessary detail sheet layers.
- Because sheet layer viewports can be easily copied and pasted, duplicate the floorplan viewports, and then copy them to the detail sheet layer. They can be rescaled and cropped to create plan detail drawings.
- When annotating a sheet layer viewport, keep in mind that “annotations” refer to any 2D element, including 2D detail objects.

Once the sheet layers and viewports have been defined, use the tools from the Detailing tool set to refine the detail drawing with either 2D or 3D detail objects. These tools can significantly reduce the amount of time necessary to make an accurate representation of an object. Detailing tool objects can be used in conjunction with other objects, such as groups, symbols, rectangles, and polylines. The following illustration shows several examples of each, used in conjunction to create a fully-developed detail form.



Creating an Interior Elevation
Creating Stipple Objects
Creating Linear Material Details
Creating Repetitive Unit Details
Creating Chain Extrude Objects
Creating Holes
Keyways
Screw Threads
Inserting Fastener Objects Using the Fastener Tool
Bolts
Nuts
Screws
Pins
Retaining Rings and Washers
Rivets
Gears
Pulleys
Sprockets
Roller Chains
Shafts
Keys

Bearings
Springs
Structural Shapes and Details

Creating an Interior Elevation

To create an interior elevation using the **Convert Copy to Lines** command:

1. From the floor plan, select the wall for creating the elevation. If it has millwork or fixtures attached to or adjacent to it, ensure that they are selected.

   Fixtures inserted in (attached to) the wall are automatically selected.

2. Select the view from which to create the elevation. For example, to view the wall from the right side of the plan, select **View > Standard Views > Right**.
3. Select **Modify > Convert > Convert Copy to Lines** and choose **Hidden Line Rendering**.
4. Select **View > Rendering > Hidden Line**. A hidden line group of the selection is created and selected. Select **Edit > Cut**.
5. Select the detail sheet layer from the **Layers** menu and select **Edit > Paste**. Move the newly created interior elevation to the desired location on the detail sheet layer and add graphics, notes, and dimensions as required.

   Alternatively, interior elevations can also be created by creating a section viewport of the interior with a finite depth range (Vectorworks Design Series required; see “Creating Section Viewports” on page 1624) or by creating a viewport with hidden line rendering and annotate the elements with the inner or outer boundary mode of the 2D Polygon tool (see “Creating Boundary Polygons in a Hidden Line Rendered Viewport” on page 306).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Convert Copy to Lines

D Creating Stipple Objects

The **Stipple** tool creates a random pattern (different shapes, sizes, and, optionally, colors) within a defined boundary, which resembles a hand-drawn shaded effect. To create a stipple object, either use the **Stipple** tool, or draw a polyline and then run the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

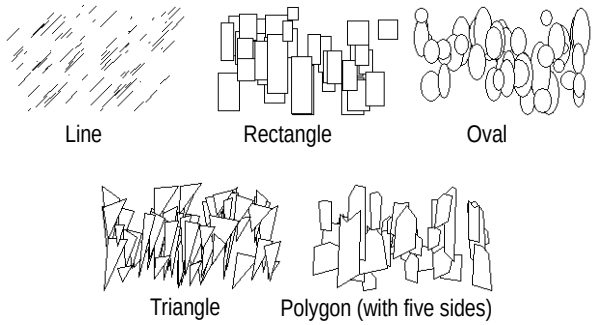
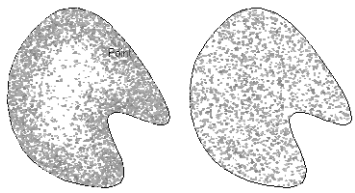
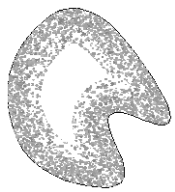
 To create a stipple object:

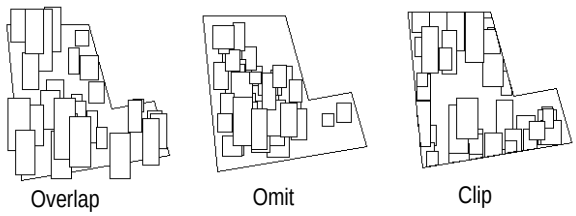
1. Click the **Stipple** tool from the Dims/Notes tool set.
2. Click **Preferences** from the Tool bar to specify or change any default **Stipple** tool parameters.

   The Define Stipple Pattern dialog box opens. Up to two different shapes can be specified to define the stipple pattern.

Click to show/hide the parameters.

Parameter	Description
Shape 1/Shape 2	Click on the appropriate tab to define two different shapes, sizes, aspect ratios, and shape percentages
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

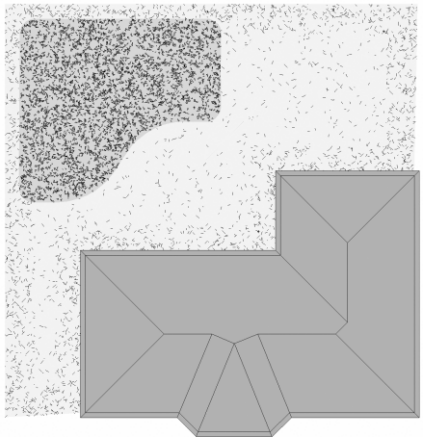
Parameter	Description
Shape type	Select the stipple fill shape. For the polygon shape, specify the number of sides. 
Minimum/Maximum Size	Enter the minimum/maximum size of the stipple pattern
Max Aspect Ratio	Enter the maximum aspect ratio of the stipple shapes (enter 1 to display square aspects)
Mix Percent	Enter the fill shape mixture percentage. The percentage defines the contribution of each shape to the stipple pattern, up to 100% (e.g. enter 60% for shape one and shape two automatically displays 40%). If shape one is assigned 100%, then shape two cannot be defined.
Stipple Density	
Low/High	Adjust the density of the stipple fill shapes by dragging the slider along the Stipple Density slider or by entering a numeric value between 1and 200 in the box below the Stipple Density slider
Color Stipple Shapes	Randomly fills the stipple shapes with colors in the specified color range
Color Range/to	Click the color boxes to select the desired color ranges
Fade Stipple from Edge	Creates a stipple that fades in from the edge of the stippled shape 
Width of Stipple	Enter the width of the faded area in page dimensions
Void Stipple Center	Leaves the center of the stipple unshaded 

Parameter	Description
Edge Treatment	Determines how the stipple shapes are drawn at the edge of the stippled area 
Randomly Rotate Shapes	Select to randomly rotate the stipple shapes. De-select to speed drawing when using fill shapes that do not require rotation, such as ovals with an aspect ratio of 1.
Use World Dimension Units	Select to use real-world dimensions; otherwise, the entries are made in page dimensions (the size the elements display when printed)

Both the **Color Stipple Shapes** and **Fade Stipple from Edge** options are processor-intensive actions and can significantly increase stipple regeneration time.

3. Click **OK**.
4. Click on the appropriate mode in the Tool bar to select the boundary creation method of the stipple object.

   For more information on the **Polyline** tool modes, see “Creating Polylines” on page 298.
5. Click to set the stipple object’s start point.
6. Click to set the end of the segment and the beginning of the next. Continue drawing segments in this manner until the stipple object is complete.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing the Stipple Object Settings

Editing the Stipple Object Settings

The parameters can be edited for selected stipple objects through the **Stipple Settings** button on the Shape tab of the Object Info palette (the parameters are described in “Creating Stipple Objects” on page 1370). To modify default stipple settings, click the **Preferences** button on the Tool bar.

Reshaping the Stipple Object

Double-click a stipple object to activate the **Reshape** tool. Select the object handles to reshape the stipple object boundary. For more information, see “Reshaping Objects” on page 1043.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Saving the Stipple Object Settings

Once the stipple object is set to the desired appearance, the settings can be saved for future use or importing into other files, by saving the stipple object. When inserted from the Resource Browser, all the stipple object settings are preset.

To save the selected stipple object:

1. Select a stipple object.
2. In the Object Info palette, click **Save Stipple**.

   The Enter String dialog box opens.
3. Enter a unique name.
4. Click **OK**.

   The stipple object is saved in the Stipples symbol folder in the Resource Browser.
5. To use a saved stipple pattern, double-click it in the Resource Browser and begin drawing in Polyline mode. For more information, see “Inserting Symbols” on page 242.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Stipple Objects

AL Creating Linear Material Details

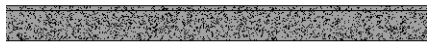
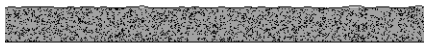
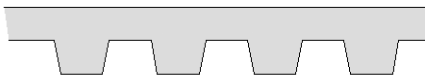
The **Linear Material** tool draws a standard representation of sheet building materials (such as plywood, gypsum board, and stucco) along a path. To create linear material details, either use the **Linear Material** tool, or draw a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

 To draw linear material details:

1. Click the **Linear Material** tool from the Detailing tool set.
2. If desired, set the linear material attributes in the Attributes palette (fill style, pen style, and line thickness). The linear material attributes can also be specified after creation.
3. Click to begin drawing the linear material detail polyline. Click the mouse at the start point to complete a closed polyline, or double-click the mouse to create an open polyline. For more information on polylines, see “Polyline Tool” on page 298.
4. The linear material parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Offset	Sets the distance of the object baseline from the drawn path
Type	Select the kind of linear material to draw; available parameters depend on the type of linear material selected
Board (generic)	Creates two polyline edges; the space between the edges can be specified with a fill color from the Attributes palette 
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Gyp. Bd. (detail)	Creates a board object with an additional offset line and fills the space with a stipple pattern 
Plaster/Stucco	Draws an irregular top line for the object 
Plaster/Stucco (detail)	Draws an irregular top line for the object, and fills the space with a stipple pattern 
Plywood/OSB	Creates a board object with evenly spaced parallel lines 
Plywood/OSB (detail)	Creates a plywood object with additional diagonal slashes 
Built-up Roof	Draws a filled band over a coarse, oval-shaped stipple pattern 
Corrugated Deck	Draws a zig-zag deck pattern 
Corrugated Deck w/fill	Creates a corrugated deck and fills the depth of the deck 
Corrugated Deck w/fill (detail)	Creates a corrugated deck, fills the depth of the deck, and adds a stipple pattern 
Thickness	Specifies the thickness of the linear material
Pitch	For corrugated deck details, determines the corrugation spacing
Corrugation Depth	For corrugated deck details, determines the corrugation height
Closure	Specifies which sections of the linear material object should have a closed end (does not apply to Built-up Roof details)

Parameter	Description
Density	For details with stipple patterns, sets the stipple density
Vertex parameters	Edits the linear detail path vertices; see “Editing Vertex-Based Objects” on page 1002

Linear material details with stipple patterns may require a significant amount of time to draw.

Create a plug-in symbol from a linear material object (see “Creating New Symbols” on page 239) with **Convert to Plug-in** selected. Activate the symbol in the Resource Browser to draw the linear material detail with the saved symbol parameters.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

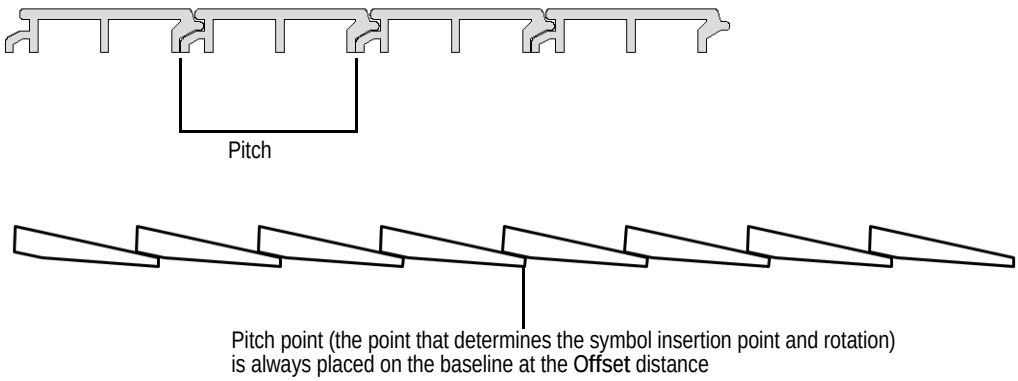
Creating Repetitive Unit Details

A L Creating Repetitive Unit Details

The **Repetitive Unit** tool draws world-scale symbols along a path, creating repetitive elements such as masonry units, shingles, and siding. To create repetitive unit details, either use the **Repetitive Unit** tool, or draw a polyline and then select the **Create Objects from Shapes** command (see “Creating Objects from Shapes” on page 277).

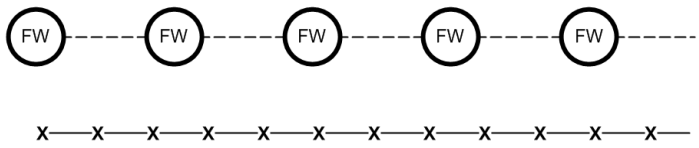
To draw repetitive unit details:

1. Click the **Repetitive Unit** tool from the Detailing tool set.
2. Click **Preferences** from the Tool bar to select the detail symbol. The detail symbol can also be specified after creation.
3. Click to begin drawing the detail polyline. Click the mouse at the start point to complete a closed polyline, or double-click the mouse to create an open polyline. For more information on polylines, see “Polyline Tool” on page 298.
4. Click **Choose Symbol** from the Object Info palette to select the unit to repeat.



Each unit is always drawn in its entirety; the last unit is not clipped, even if it exceeds the path drawn. The repetitive unit offset path is drawn in the object’s assigned pen style (the path always has a fill style of None).

The **Repetitive Unit** tool can create lines with custom symbols made from text and/or shapes. The **Pitch** determines the spacing; set the pen attributes from the Attributes palette.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



The repetitive unit parameters can be edited in the Object Info palette.

Click to show/hide the parameters.

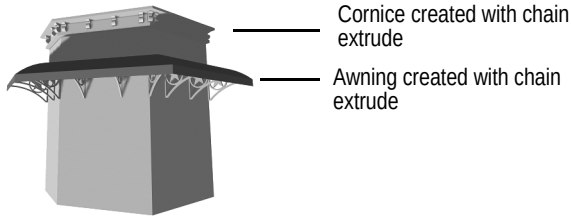
Parameter	Description
Offset	Sets the distance of the object baseline from the drawn path; the symbol is rotated so that its first point (its "pitch point") is always placed on the baseline at the offset distance
Pitch	Sets the distance from the start of one unit to the start of the next unit
Use Symbol Pitch	Sets the pitch according to the symbol definition; deselect to scale the pitch manually
Use Vertical Pitch	Rotates the symbol 90 degrees counter-clockwise and sets the pitch vertically; this is useful for symbols that are to be stacked vertically rather than arrayed vertically, as they can be drawn in a more natural way
First Pitch Differs	Allows a different pitch to be specified for the first unit
First Pitch	Specifies the pitch for the first unit
Flip Vertically	Flips the unit symbol vertically
Flip Horizontally	Flips the unit symbol horizontally
Choose Symbol	Opens the Choose Symbol dialog box. Select the unit **Category**, and then select the unit symbol from either the default content or the current file's content (see "Resource Libraries" on page 219); the selected symbol is imported into the file and appears in the Resource Browser.
Vertex Parameters	Edits the repetitive unit detail path vertices; see "Editing Vertex-Based Objects" on page 1002

Create a plug-in symbol from a repetitive unit object (see "Creating New Symbols" on page 239) with **Convert to Plug-in** selected. Activate the symbol in the Resource Browser to draw the repetitive unit with the saved symbol parameters.

Creating Linear Material Details

Creating Chain Extrude Objects

The **Chain Extrude** tool automatically creates extruded objects with both continuous and repetitive elements. Architectural features such as an awning contain a continuous element (the awning) and intermittently repetitive elements (the supports). By specifying the polygon or polylines to be extruded, and then indicating the path of the extrusion, the object(s) are automatically extruded along the path.



Creating the Profile Objects
Creating the Chain Extrude Object
Chain Extrude Properties

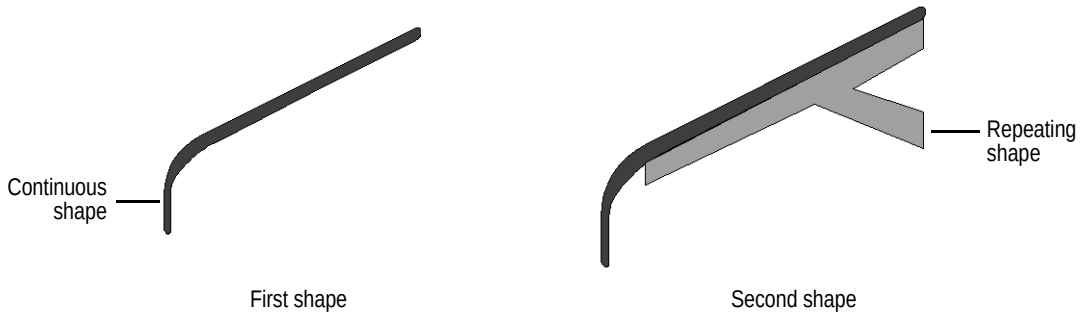
Creating the Profile Objects

The profile objects determine the shape of the chain extrude object. The profile objects can consist of up to three polygons/polylines or up to two polygons/polylines and one locus. For users of the Vectorworks Architect and Landmark products, profile objects are contained in the wall fixtures and accessories file of the [Vectorworks]\Libraries folder (see “Resource Libraries” on page 219).

To create the profile objects:

1. Create the continuous shape first. The **Chain Extrude** tool considers the first, or back, polygon/polyline to be the continuous shape, and extrudes it along the path polygon.
2. Create the second shape; the front object is considered to be the repeating or intermittent object.

   A second shape is not required.



   The reference location of the chain extrude relative to the path is the upper right or lower right corner of the shape’s bounding box. This location can be changed by including a locus in the profile group. The locus becomes the reference point for the path polygon in the profile group. This is useful if the bounding box of the repeating shape extends beyond the bounding box of the continuous shape.
3. Apply fill colors to the profile objects through the Attributes palette. The colors are applied to the continuous and repeating shapes in 3D.

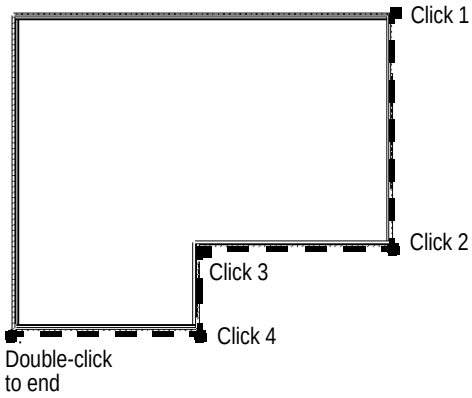
Creating the Chain Extrude Object

Creating the Chain Extrude Object

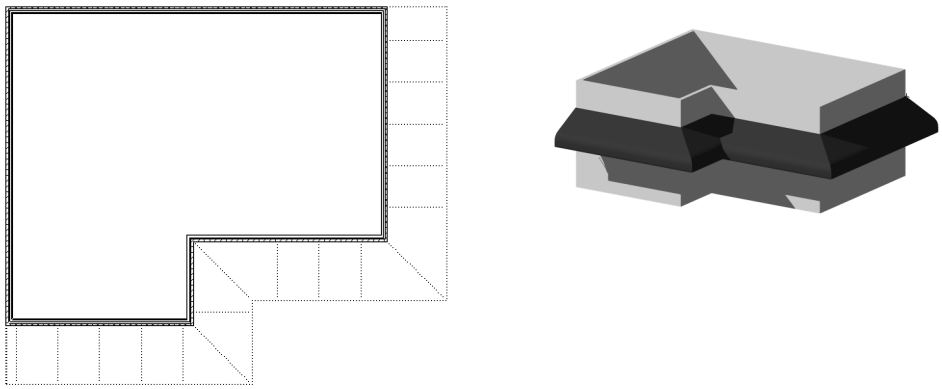
Once the continuous and, if any, repeating shapes have been defined, specify the path for the chain extrude with the **Chain Extrude** tool.

To create the chain extrude object:

1. Select the continuous and, if any, repeating shapes.
2. Click the **Chain Extrude** tool from the Building Shell tool set.
3. Draw the chain extrude path polyline.



4. The chain extrude object is created based on the path drawn.



In 2D, the chain extrude object is displayed with dashed single lines

Chain Extrude Properties

Chain Extrude Properties

The chain extrude object parameters can be edited in the Object Info palette.

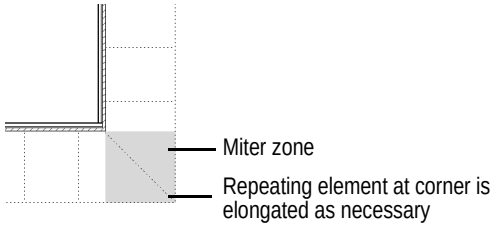
Click to show/hide the parameters.

Parameter	Description
Draw 3D	Select to display the chain extrude object in 3D view; deselect to display the object only in 2D view
Flip	Select to flip the orientation of the chain extrude object (for example, toward the inside of a room for a crown molding, instead of the outside of a building for a cornice)
Center Ribs	Select to draw the repeating element centered along the edges of the path
Corner Ribs	Select to draw the repeating element at the corners of the path

Parameter	Description
Vertical Datum	Specifies the reference point for the chain extrude object
Thickness	Specifies the thickness of the repeating elements of the object
Spacing	Indicates the spacing distance between the repeating elements of the object
Vertex parameters	Edits the chain extrude path vertices; see “Editing Vertex-Based Objects” on page 1002

The placement of the repeating elements of the chain extrude object depends on the path created by the **Chain Extrude** tool. In particular, special considerations apply to the “miter zones” at the corners of the path:

- No repeating elements perpendicular to the building face will be drawn in miter zones
- The miter zones are excluded from the length calculations of the sides (for spacing calculations, for example)
- Repeating elements at corners, if drawn, bisect the angle of the corner and are elongated accordingly



The chain extrude path can be reshaped with the **Reshape** tool to add, subtract, and change vertices. The continuous and repetitive elements are automatically adjusted to fit the new shape. The path polyline can be edited by selecting **Modify > Edit Chain Extrude**, and selecting **Path**, or by selecting **Edit** or **Edit Path** from the context menu (see “Object Editing Mode” on page 1004).

Creating Chain Extrude Objects

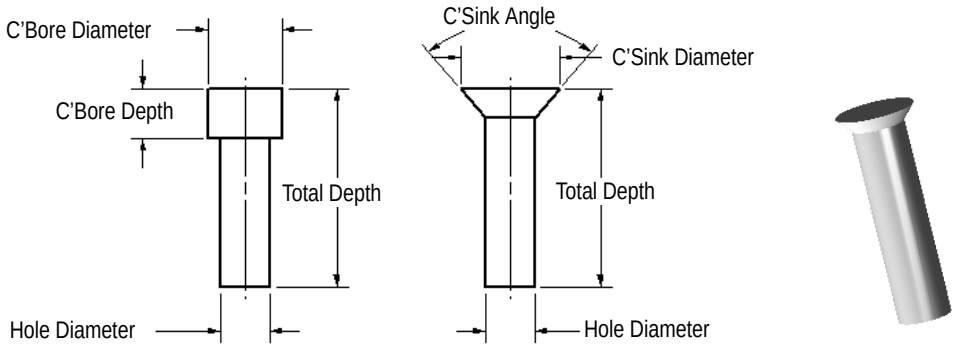
Creating Holes

Drilled Holes

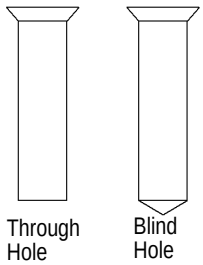
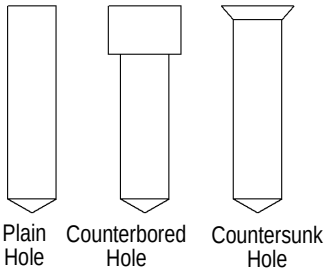
Both 2D and 3D drilled holes are available. The 3D drilled hole is a solid object that can be subtracted from any other solid object to create accurate 3D models of molds, dies, and many other machine parts (see “Converting to Generic Solids” on page 373).

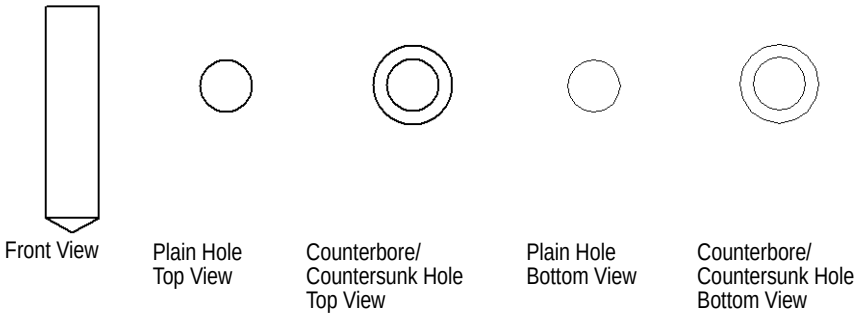
To insert a drilled hole:

1. Click the **Hole - Drilled** tool or **Hole - Drilled - 3D** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object’s rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object’s properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Hole Diameter	Enter the hole diameter
Total Depth	Enter the depth of the hole
Hole Type	Select one of the hole types 
Drill Point Angle	For Blind Hole types, indicate the angle of the drill point at the bottom of the hole
Hole Style	Select the hole style 
Diameter	For counterbored or countersunk hole styles, enter the diameter of the counterbore or countersink
Depth	For counterbored hole styles, specify the depth of the counterbore
Angle (deg.)	For countersunk hole styles, specify the angle of the countersink

Parameter	Description
View	Select the 2D view 
Show Center Line	Select to draw the 2D drilled hole with center line(s)

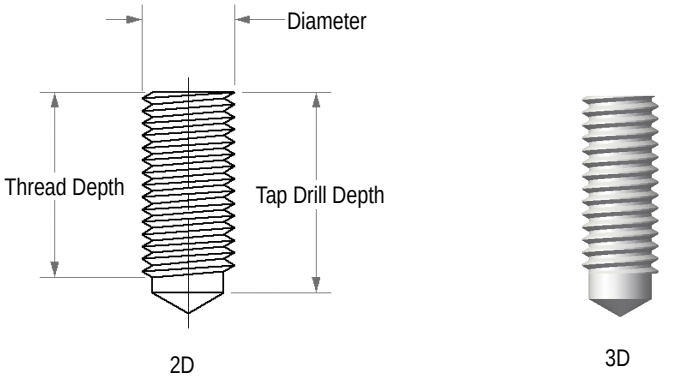
Tapped Holes

Tapped Holes

Both 2D and 3D tapped holes are available. The 3D tapped hole is a solid object that can be subtracted from any other solid object to create accurate 3D models of molds, dies, and many other machine parts (see “Converting to Generic Solids” on page 373).

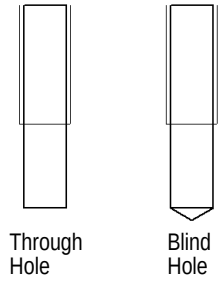
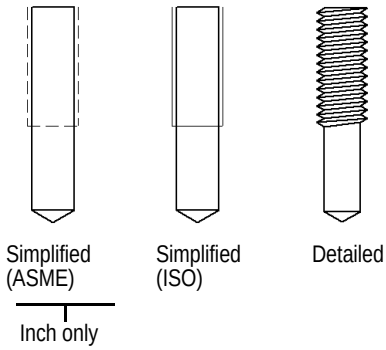
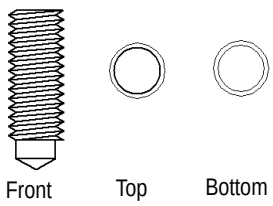
To insert a tapped hole:

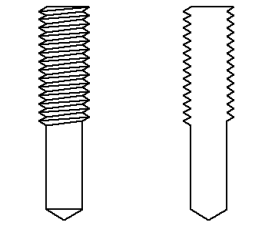
1. Click the desired **Hole - Tapped** inch or metric tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object’s rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object’s properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Screw Thread	Select the screw thread size in diameter/threads per inch (imperial) or diameter/thread pitch (metric)

Parameter	Description
Tap Drill Size (Ref.)	Displays the tap drill size that corresponds to the selected screw thread
Hole Type	Select one of the hole types 
Thread Depth	Specify the thread depth
Tap Drill Depth	Enter the tap drill depth value
Drill Point Angle	For Blind Hole types, enter the drill point angle
Use Custom Size	Select to use custom diameter, thread pitch, and tap drill diameter values
Diameter	When **Use Custom Size** is selected, specify the diameter value
Pitch	When **Use Custom Size** is selected, specify the thread pitch value
Tap Drill Dia.	When **Use Custom Size** is selected, specify the tap drill diameter value
Thread Style	Select the 2D object thread style 
View	Select the 2D object view 

Parameter	Description
Show Profile Only	Select to show just the outline of the 2D object thread. Available when using **Detailed** thread style only. 
Show Center Line	Select to draw the 2D tapped hole object with center line(s)
Show Threads	Select to draw the 3D tapped hole object with threads

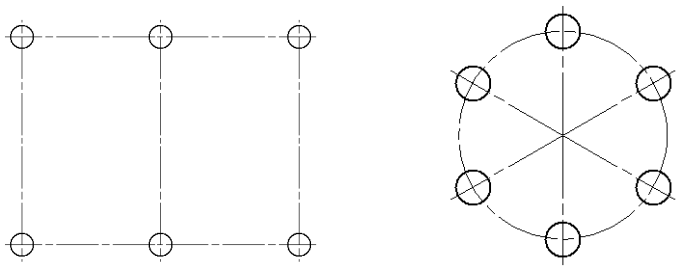
Drilled Holes

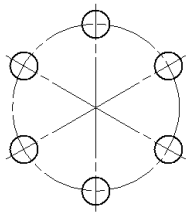
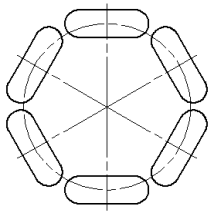
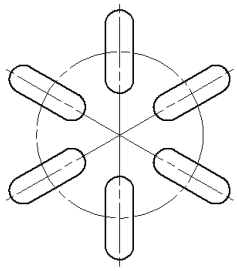
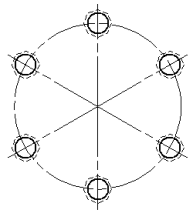
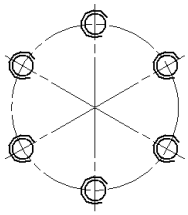
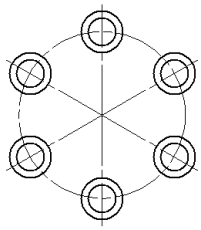
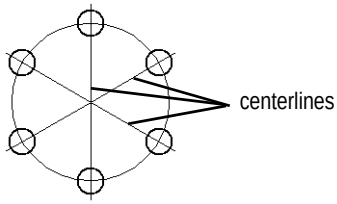
D Hole Pattern

To insert a hole pattern:

1. Click the **Hole Pattern** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object's properties and click **OK**.

Click to show/hide the parameters.

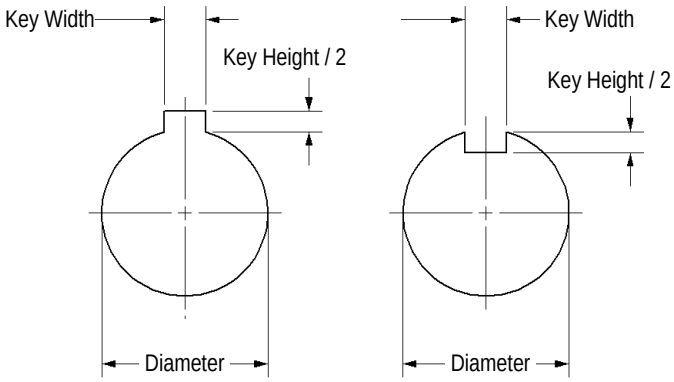
Parameter	Description
Pattern	Select a hole pattern 
Number of Rows	For rectangular hole patterns, enter the number of rows
Vertical Spacing	For rectangular hole patterns, enter the vertical spacing between the rows
Number of Columns	For rectangular hole patterns, enter the number of columns
Horizontal Spacing	For rectangular hole patterns, enter the horizontal spacing between the columns
Number of Holes	For bolt circle patterns, enter the number of holes

Parameter	Description
Bolt Circle Dia.	For bolt circle patterns, enter the diameter
Hole Type	Select the type of holes  Round  Slotted - Horiz  Slotted - Vert  Tapped Hole - ASME Style  Tapped Hole - ISO Style
Hole Size	For round and tapped hole types, enter the hole size
Slot Width	For slotted hole types, enter the slot width
Slot Length	For slotted hole types, enter the slot length
Counterbore/ Countersunk	Select to draw counterbore or countersunk holes 
Diameter	Indicates the diameter of the counterbore or countersunk holes
Show Centerlines	Select to display the centerlines on the pattern 
View	Select the hole pattern view

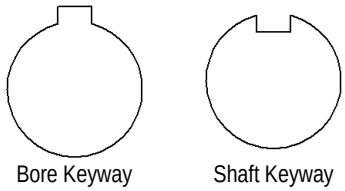
D Keyways

To insert a keyway:

1. Click the **Keyway** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Shaft/Bore Diameter	Enter the diameter of the shaft or bore
Type of Keyway	Select one of the keyway types 
Key Size Standard	Select inch or metric for the key size standard
Type of Key	Select a square, rectangular, or woodruff key shape
Use Recommended Key Size	Select to use the recommended ASME/ISO size based on the diameter and key type; deselect to specify custom width and height values
Key Width/Height	For custom keyway sizes, enter the width and height values
Show Center Line	Select to draw the keyway with center lines

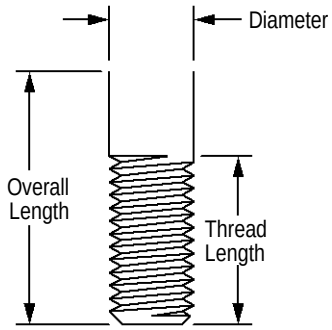
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Keys
Woodruff Keys
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

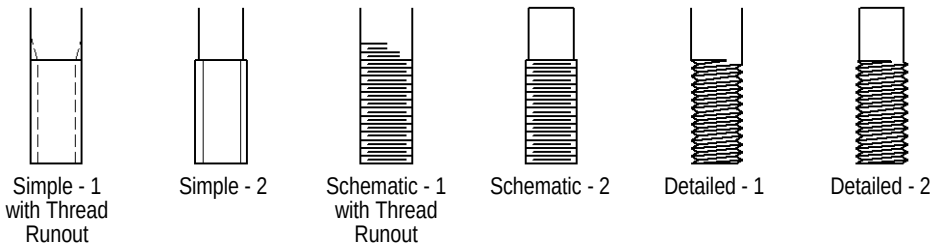
D Screw Threads

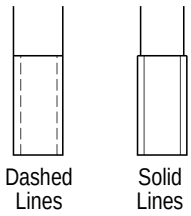
To insert a screw thread:

1. Click the **Screw Threads** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Diameter	Enter the thread diameter
Overall Length	Specify the overall length of the screw thread
Thread Length	Specify the length of the threads only
Thread Pitch	Indicate the thread pitch
Fillet Radius	Specify the fillet radius
Thread Style	Select one of the thread display styles. For Style 1, the shoulder diameter is equal to the thread diameter; for Style 2, the shoulder diameter is equal to the pitch diameter. 
Thread Bottom	Select flat or chamfered

Parameter	Description
Show Threads As	For Simple thread styles, select dashed lines or solid lines 
Show Thread Runout	For Simple-1 and Schematic-1 styles, select to display the thread runout
Show Center Line	Select to draw the screw threads with center line(s)

Inserting Fastener Objects Using the Fastener Tool

Several types of fastener objects can be inserted by clicking the **Fastener** tool from the Fasteners tool set and specifying object parameters from the Tool bar.

The following modes are available.

Mode	Description
Screw or Screw/ Washer/Nut Combination	Inserts a screw and nut object which can be composed of a variety of screw, washer, and/or nut components; hex bolts, square bolts, hex screws, cap screws, and machine screws can all be inserted using this tool mode
Nut	Inserts a nut object, including hex, hex jam, slotted hex, and square nuts
Plain Washer	Inserts a plain washer object
Lock Washer	Inserts a lock washer object
Shoulder Screw	Inserts a shoulder screw object
Set Screw	Inserts a set screw object
2D Front View	Inserts the selected item as a 2D object in front view
2D Side View	Inserts the selected item as a 2D object in side view
2D Top View	Inserts the selected item as a 2D object in top view
2D Bottom View	Inserts the selected item as a 2D object in bottom view
3D Object	Inserts the selected item as a 3D object
ASME Inch	Inserts the selected object using ASME inch standards
ASME Metric	Inserts the selected object using ASME metric standards
ISO	Inserts the selected object using ISO standards
DIN	Inserts the selected object using DIN standards
Preferences	Opens the Preferences dialog box for specification of additional object parameters

To insert a fastener object using the Tool bar:

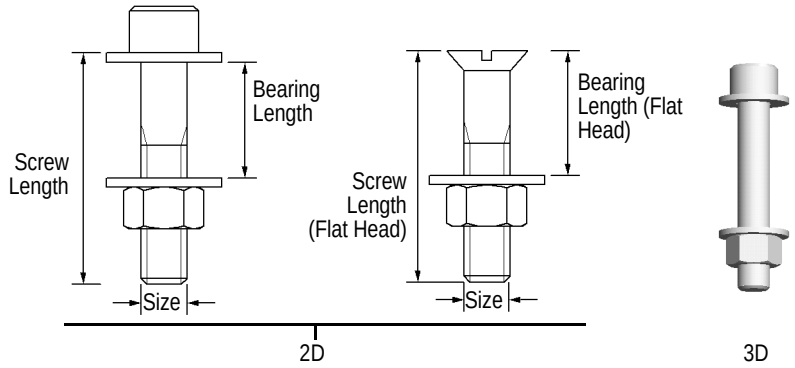
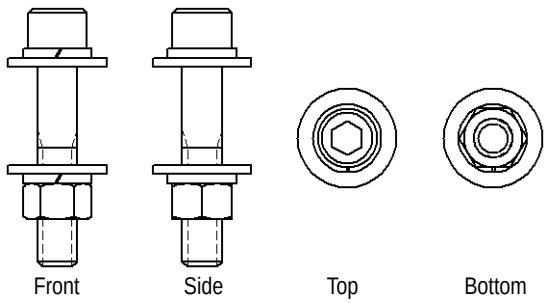
1. Click the **Fastener** tool from the Fasteners tool set and click the desired fastener object button in the Tool bar.
2. Click the Tool bar button to insert either a 2D object in a specific view or a 3D object.

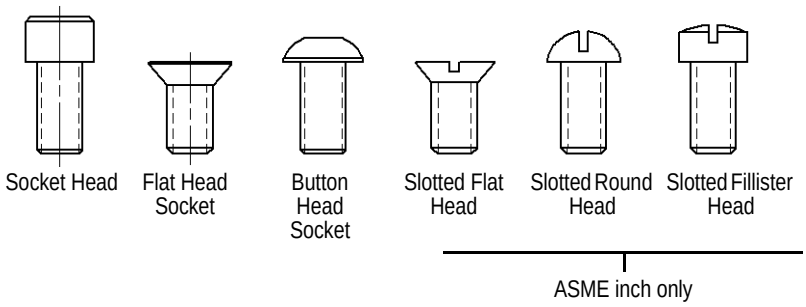
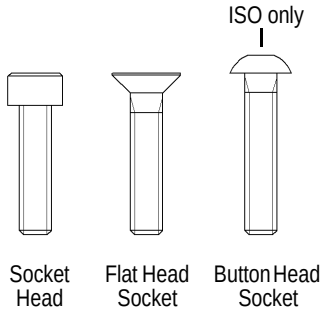
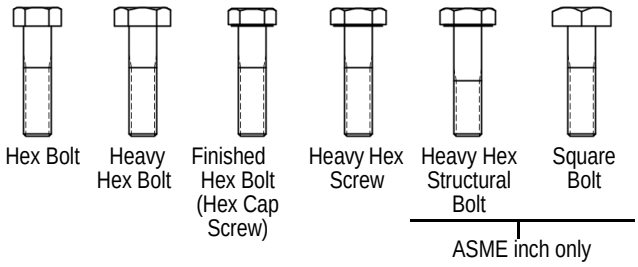
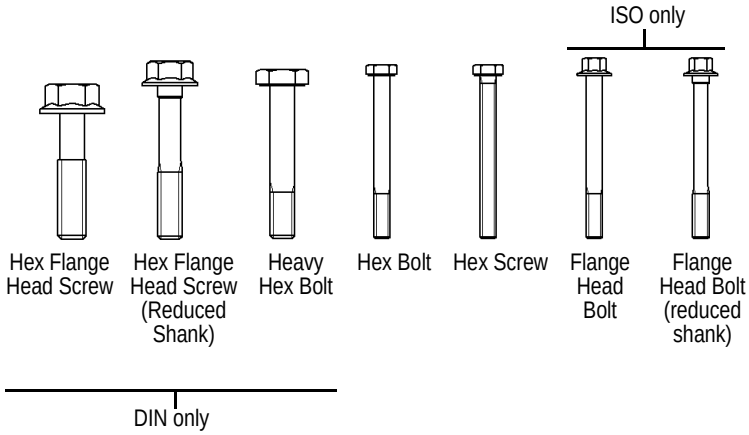
   For objects that do not have bottom view (because it is the same as the top view), the top view is drawn when either the 2D Top View or 2D Bottom View mode is selected. The same applies to the front and side view.
3. Click the Tool bar button to insert the object using ASME (inch or metric), ISO, or DIN standards.
4. Click **Preferences** on the Tool bar to specify additional object parameters before placing the object in the drawing. Parameters vary based on the object selected for insertion, and show the available variations for a specified object size. Additional parameters display in the Object Info palette. Refer to the following table for detailed object parameter information.
5. If inserting a linear object (such as a bolt, screw, or screw and nut object), click in the drawing area to set the start point, drag to set the object's length, and click again to set the object's end point. For all other objects (such as washers and nuts), click in the drawing area to place the selected point object.

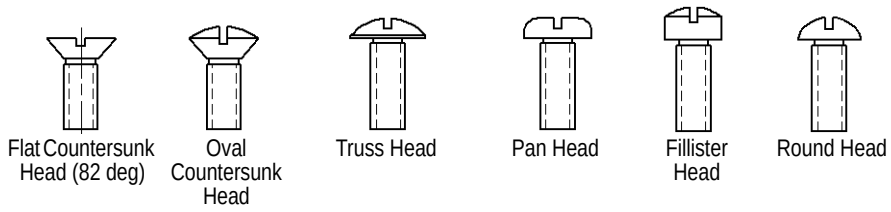
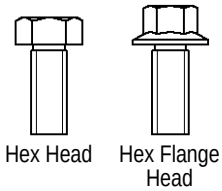
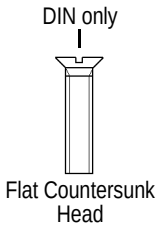
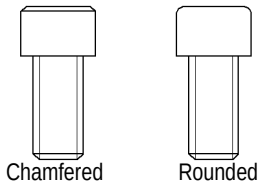
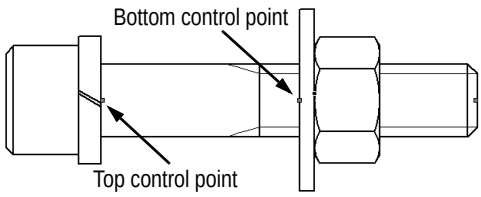
After placement, fastener objects can be edited in the Object Info palette.

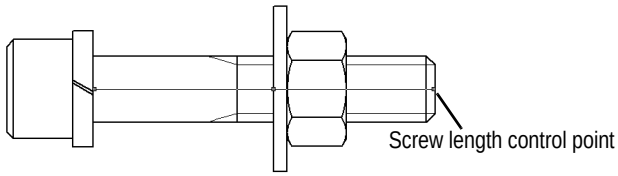
Screw or Screw/Washer/Nut Combination Parameters

Click to show/hide the parameters.

Parameter	Description
Screw or Screw/ Washer/Nut Combination	
View	Select the 2D view 
Size	Select the screw size (ASME inch), or the diameter/thread pitch (ASME metric, DIN, and ISO)
Configuration	Opens the Configuration dialog box to select size, screw/bolt, washer, and nut type. See the following pages for illustrations of the available lock washers, washers, and nuts. The list of screw/bolt, washer, and nut types varies depending on the selected size. Not all types are available for all sizes.

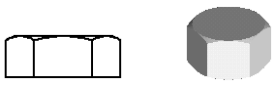
Parameter	Description
Screw / Bolt Type	Specify the screw/bolt type
Cap screw ASME inch and metric	
Cap screw DIN and ISO	
Hex bolt/ screw ASME inch and metric	
Hex bolt/ screw DIN and ISO	

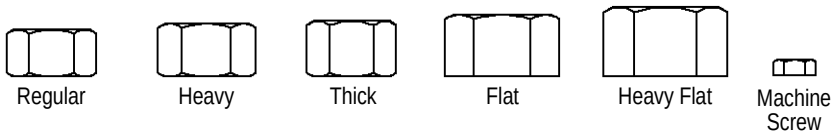
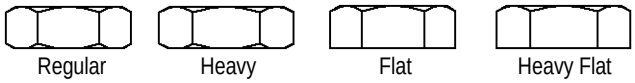
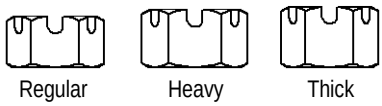
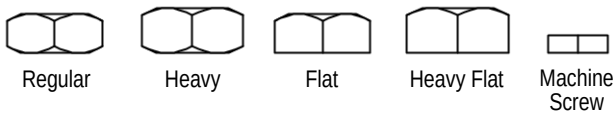
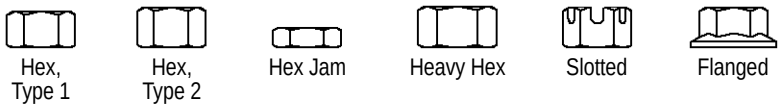
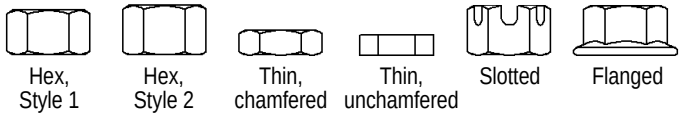
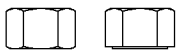
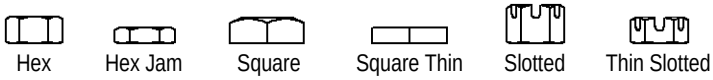
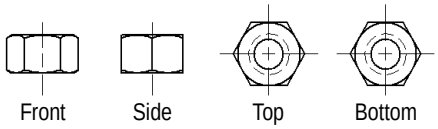
Parameter	Description
Machine screw ASME inch	
Machine screw ASME metric	
Machine screw DIN and ISO	
Head Style (ASME metric, ISO, and DIN only)	For cap screws (ASME metric, ISO, and DIN), specify the head style 
Top/Bottom Lock Washer	Select to draw a top and/or bottom lock washer; several lock washer types are available
Top/Bottom Washer	Select to draw a top and/or bottom washer; several washer types are available
Nut	Select to draw a nut. For Hex, Style 1 and Hex, Style 2 nut ISO types, specify either a regular or washer-face **Nut Form**; several nut types are available
Bearing Length	Specify the bearing length or click and drag the bottom bearing length control point to define the bearing length graphically 

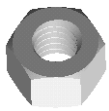
Parameter	Description
Screw Length	For a custom length value, deselect **Use Standard Length Increments** and enter the length under the screw head; alternatively, click and drag the screw length control point to define the length graphically 
Thread Pitch (ASME inch only)	For ASME inch sizes, specify Unified National Coarse or Unified National Fine threads
Thread Style	Select the 2D thread style; see “Fastener Object Thread Styles” on page 1397
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on size, per ASME, ISO, or DIN standards; deselect to enter a custom length
Adjust Screw Length Automatically	Select to automatically adjust the screw length to keep the screw threads within a nut or to prevent the bearing length from exceeding the screw length
Show Threads	Select to create the 3D screw/bolt with threads 
Draw Underhead Fillet	Select to draw a fillet under the head of the 2D hex bolt/hex screw
Show Center Line	Select to draw the 2D screw and nut with center line(s)

Nut Parameters

Click to show/hide the parameters.

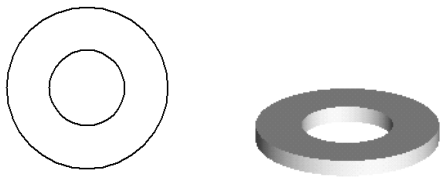
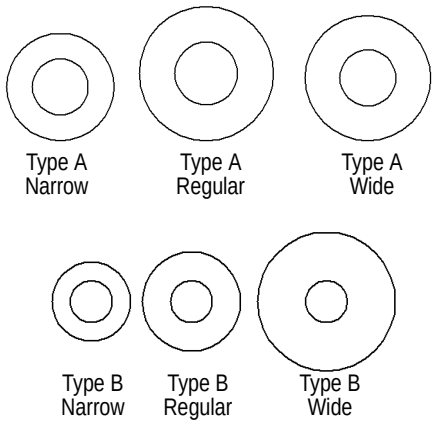
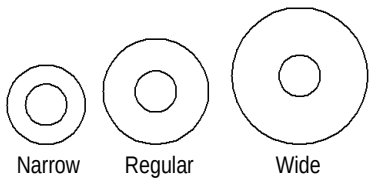
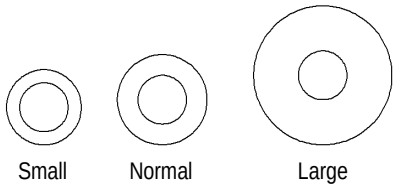
Parameter	Description
Nut	
Size	Select the nut size
Type	Specify the nut type

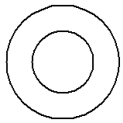
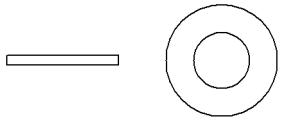
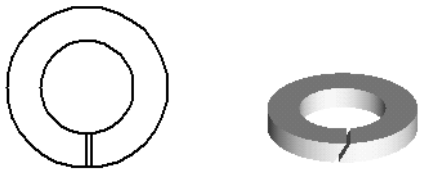
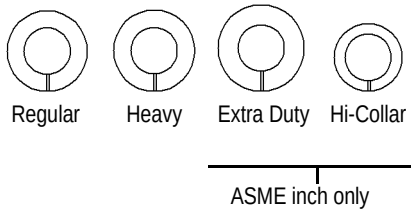
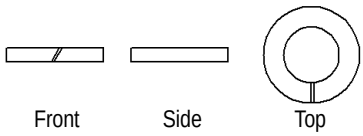
Parameter	Description
ASME inch	Hex nuts 
	Hex jam nuts 
	Slotted hex nuts 
	Square nuts 
ASME metric	
ISO	
Form	For Hex, Style 1 and Hex, Style 2 nut (ISO) objects, specify either a regular or washer-face form 
DIN	
View	Select the 2D view 
Show Center Line	Select to draw the 2D nut with center line(s)

Parameter	Description
Show Hole	Select to create the 3D nut with a center hole 
Show Threads	Select to create the 3D nut with threads 

Washer Parameters

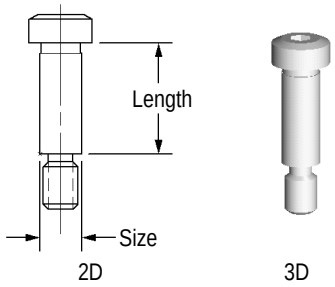
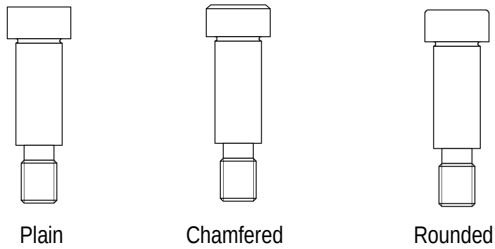
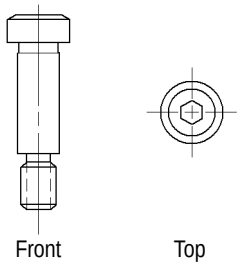
Click to show/hide the parameters.

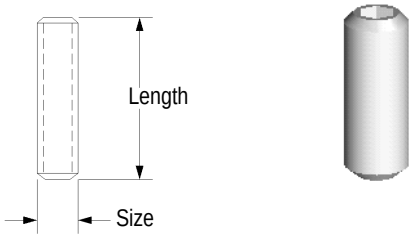
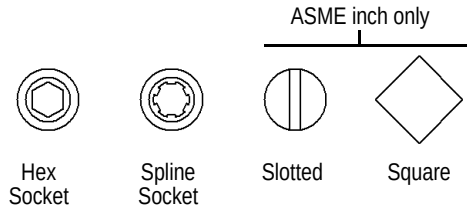
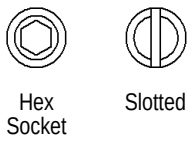
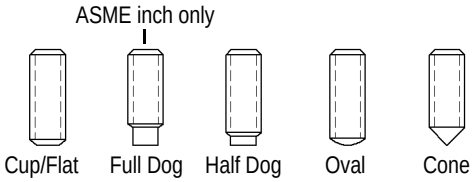
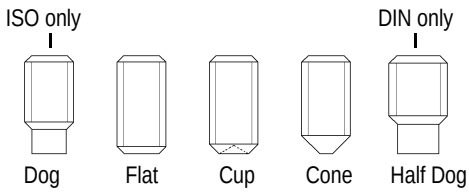
Parameter	Description
Plain Washer	
Type/Series	Specify the plain washer type
ASME inch	
ASME metric	
ISO	

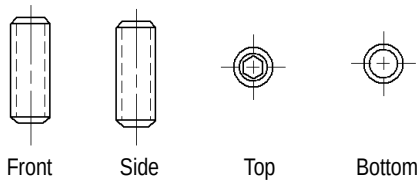
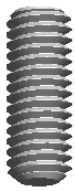
Parameter	Description
DIN	Type is not applicable for DIN objects 
Size	Select the plain washer size
View	Select the 2D view 
Show Center Line	Select to draw the 2D plain washer with center line(s)
Lock Washer	
Series (ASME inch and metric and ISO only)	Specify the lock washer series (not applicable for DIN objects) 
Size	Select the lock washer size
View	Select the 2D view 
Show Center Line	Select to draw the 2D lock washer with center line(s)

Screw Parameters

Click to show/hide the parameters.

Parameter	Description
Shoulder Screw	
Size	Select the shoulder screw size
Length	For a custom length value, deselect **Use Standard Length Increments** and enter the length under the screw head
Head Style (ISO only)	Select the head style 
Thread Style	Select the 2D thread style; see “Fastener Object Thread Styles” on page 1397
View	Select the 2D view 
Draw Underhead Fillet (ASME inch, ASME metric, and DIN only)	Select to draw a fillet under the head of the shoulder screw
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on size, per ASME, ISO, or DIN standards; deselect to enter a custom length
Show Center Line	Select to draw the 2D shoulder screw with center line(s)

Parameter	Description
Show Threads	Select to create the 3D shoulder screw with threads 
Set Screw	
Head Type	Specify the set screw head type
ASME inch and metric	
DIN and ISO	
Point Type	Specify the set screw point type
ASME inch and metric	
DIN and ISO	
Size	Select the set screw size
Length	For a custom length value, deselect **Use Standard Length Increments** and enter the length under the screw head

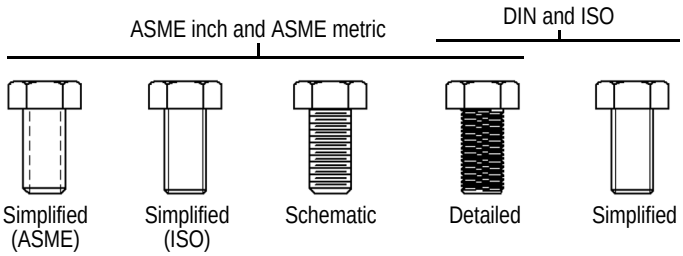
Parameter	Description
Thread Pitch (ASME inch only)	For ASME inch sizes, specify Unified National Coarse or Unified National Fine threads
Thread Style	Select the 2D thread style; see “Fastener Object Thread Styles” on page 1397
View	Select the 2D view 
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on size, per ASME, ISO, or DIN standards; deselect to enter a custom length
Show Center Line	Select to draw the 2D set screw with center line(s)
Show Threads	Select to create the 3D set screw with threads 

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Fastener Object Thread Styles

D Fastener Object Thread Styles

There are several thread types common to multiple fastener tools. The available thread types vary by the drawing standard being used.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Fastener Objects Using the Fastener Tool

A Bolts

The following objects allow the creation and application of user-defined sizes:

- bolts and nuts
- carriage bolts
- eye bolts
- J-bolts
- swing bolts
- swing eye bolts
- T-bolts
- U-bolts

See “Customizing Size Settings for Plug-in Objects” on page 1792 for instructions on establishing custom object sizes.

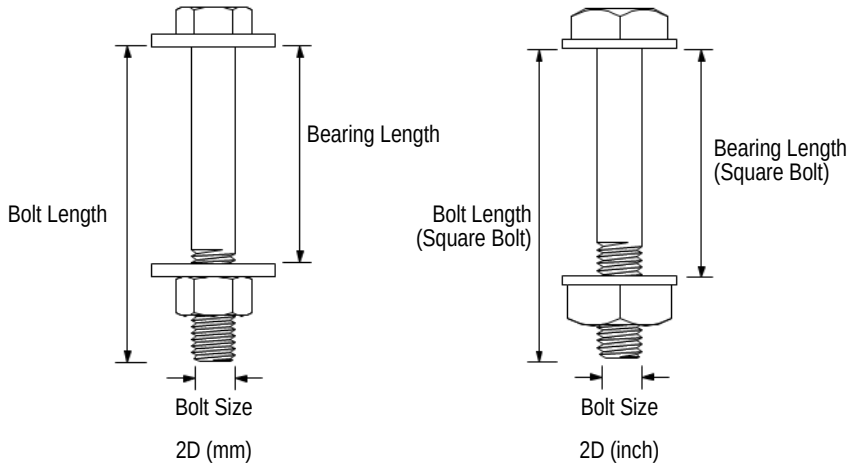
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Fastener Objects Using the Fastener Tool
Bolts and Nuts
Carriage Bolts
Eye Bolts
J-bolts
Swing Bolts
Swing Eye Bolts
T-bolts
U-bolts

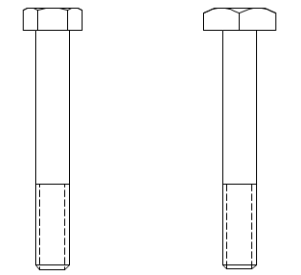
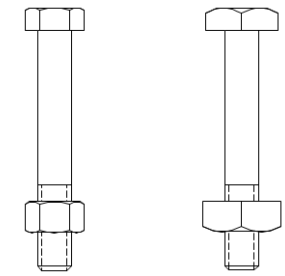
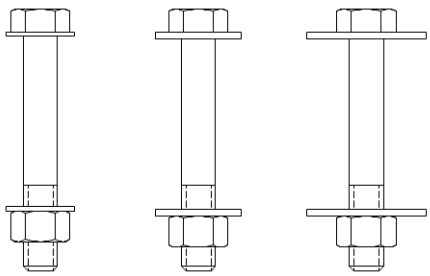
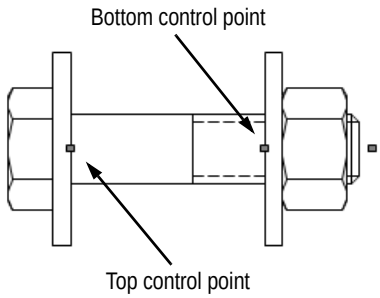
Bolts and Nuts

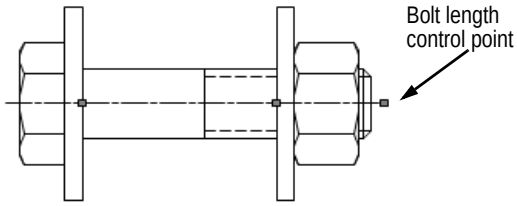
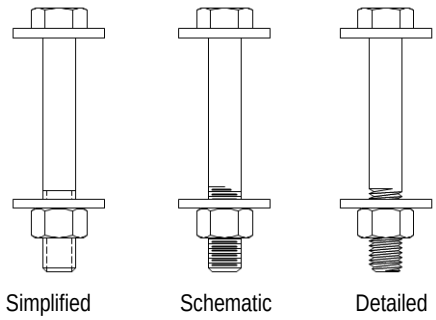
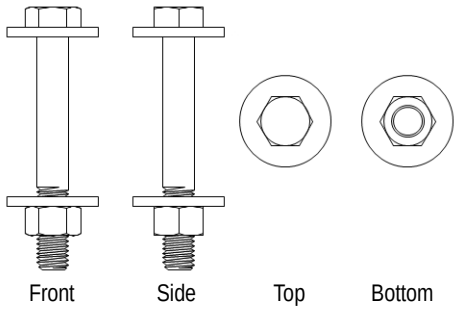
To insert a bolt and nut (mm or inch):

1. Click the **Bolt and Nut** tool from the Detailing tool set.
2. Click in the drawing area to set the start point, drag to set the object’s length, and click again to set the object’s end point. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Bolt Type (inch only)	Select the bolt type  Hex Bolt Square Bolt
Nut Type (inch only)	Select the nut type  Hex Nut Square Nut
Washer Type	Select a washer type  Narrow Regular Wide
Bolt Size	Select the bolt size
Bearing Length	Specify the bearing length or click and drag the bottom bearing length control point to define the bearing length graphically  Bottom control point Top control point

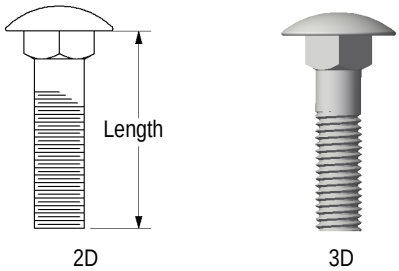
Parameter	Description
Bolt Length	For a custom length value, deselect **Use Standard Lengths** and enter the length under the bearing length; alternatively, click and drag the bolt length control point to define the length graphically 
Thread Style	Select the bolt thread style 
Top Washer/Top Lock Washer/Bottom Washer/ Bottom Lock Washer	Select to draw the bolt with top and/or bottom washers and top and/or bottom lock washers
Nut	Select to draw the bolt with a nut
View	Select the 2D view 
Use Standard Lengths	Select to automatically adjust the bolt length to the nearest increment based on size; deselect to enter a custom length
Adjust Bolt Length	Select to automatically adjust the bolt length based on the bearing length
Show Center Line	Select to draw the 2D bolt and nut with center line(s)

Bolts
Inserting Fastener Objects Using the Fastener Tool

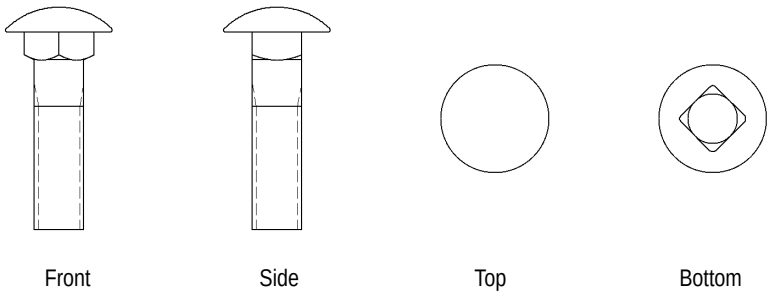
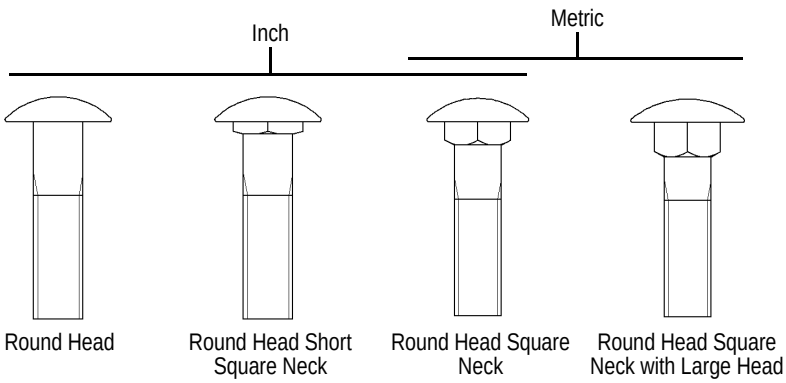
D Carriage Bolts

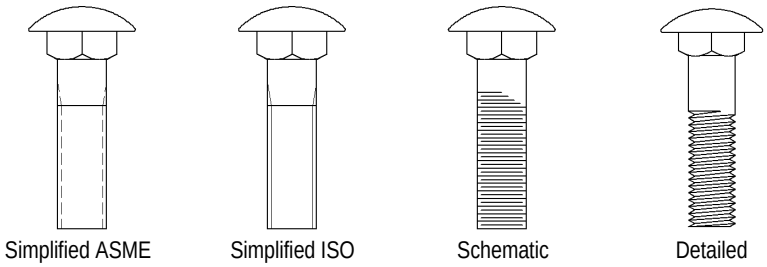
To insert a carriage bolt (round head/round head square neck bolt):

1. Click the **Carriage Bolt** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
View (2D only)	Select the 2D view 
Series	Select inch or metric to display the appropriate nominal sizes
Bolt Type	Select the carriage bolt type 
Size	Select the carriage bolt size

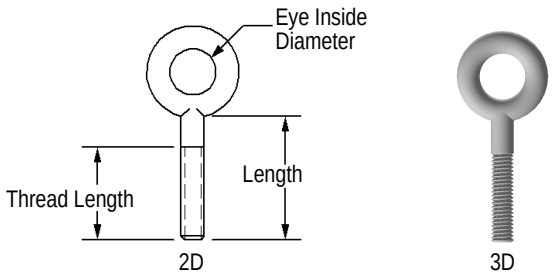
Parameter	Description
Length	For a custom length value, deselect **Use Standard Length Increments** and enter the length under the bolt head
Thread Style (2D only)	Select the thread style 
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on size, per ASME standards; deselect to enter a custom length
Show Center Line (2D only)	Select to draw the carriage bolt with center line(s)

Inserting Fastener Objects Using the Fastener Tool
Bolts

D Eye Bolts

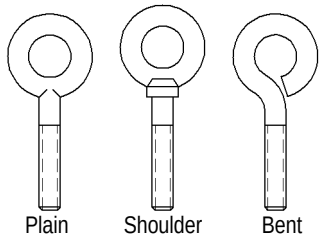
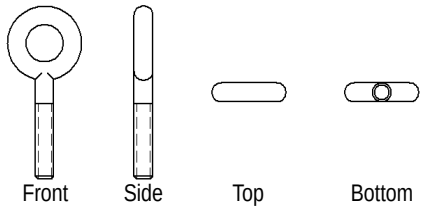
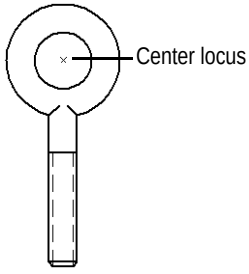
To place an eye bolt:

1. Click the **Eye Bolt** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Specify the eye bolt size in diameter/threads per inch (imperial) or diameter/thread pitch (metric)

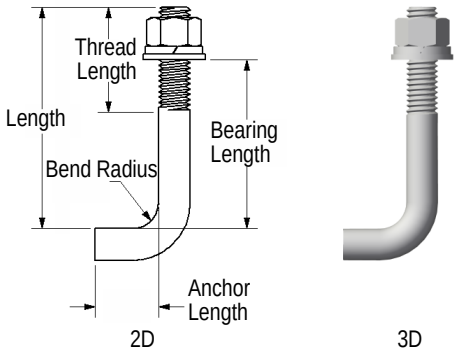
Parameter	Description
Type	Specify the eye bolt type 
Eye Inside Dia	Specify the eye bolt's inside diameter
Length	Specify the length under the bolt head
Thread Length	Specify the thread length
Thread Style (2D only)	Select the thread style; see "Fastener Object Thread Styles" on page 1397
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the eye bolt with center line(s)
Show Threads (3D only)	Select to create the eye bolt with threads 
Place Locus at Center	Select to draw the eye bolt with a center locus 

Inserting Fastener Objects Using the Fastener Tool
Bolts

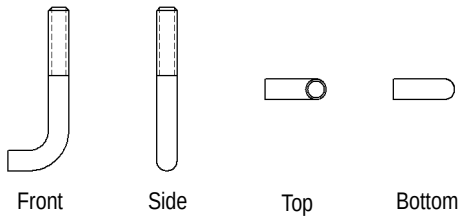
D J-bolts

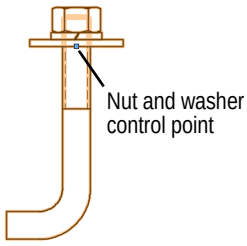
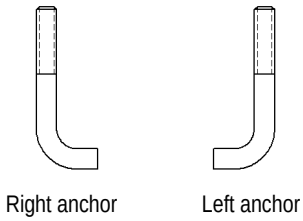
To insert a J-bolt:

1. Click the **J-Bolt** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
View (2D only)	Select the 2D view 
Size	Specify the J-bolt size in diameter/threads per inch (imperial) or diameter/thread pitch (metric)
Length	Specify the bolt length
Thread Length	Specify the thread length
Bend Radius	Specify the radius of the bend
Anchor Length	Specify the anchor length
Thread Style (2D only)	Select the thread style; see "Fastener Object Thread Styles" on page 1397

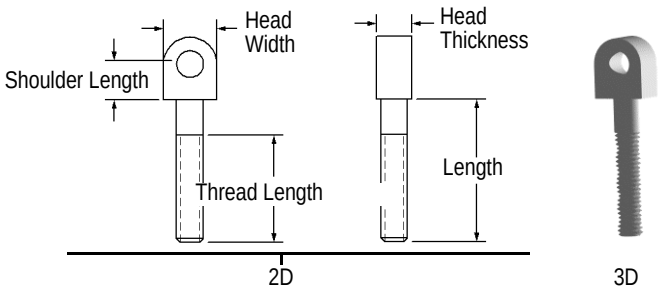
Parameter	Description
Add Nut	Select to draw the J-bolt with a hex or square nut; click the control point to adjust the nut placement graphically and click again to place the nut 
Add Lock Washer / Washer	Select to draw the J-bolt with a lock washer and/or with a narrow, regular or wide washer
Bearing Length	Specify the bearing length; if a nut or washer is added or removed, the bearing length remains constant and the position of the existing objects is adjusted accordingly (if the nut or washer is moved, the bearing length is not automatically adjusted)
Show Threads (3D only)	Select to create the J-bolt with threads 
Anchor to the right (2D only)	Select to draw the J-bolt with an anchor to the right or deselect to draw the anchor to the left 
Show Center Line (2D only)	Select to draw the J-bolt with center line(s)

Inserting Fastener Objects Using the Fastener Tool
Bolts

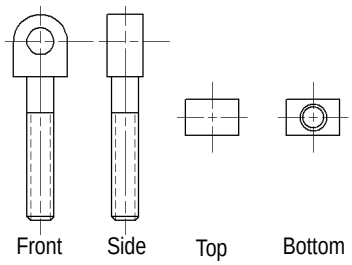
Swing Bolts

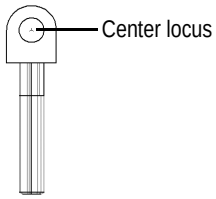
To insert a swing bolt:

1. Click the **Swing Bolt** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Specify the swing bolt size in diameter/threads per inch (imperial) or diameter/thread pitch (metric)
Head Width	Specify the swing bolt head width
Shoulder Length	Specify the swing bolt shoulder length
Head Thickness	Specify the swing bolt head thickness
Length	Specify the length under the bolt head
Thread Length	Specify the thread length
Thread Style (2D only)	Select the thread style; see “Fastener Object Thread Styles” on page 1397
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the swing bolt with center line(s)
Show Threads (3D only)	Select to create the swing bolt with threads 

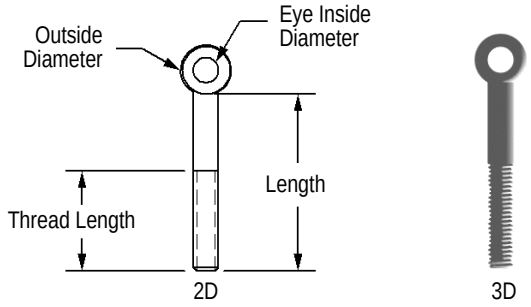
Parameter	Description
Place Locus at Center	Select to draw the swing bolt with a center locus 

Inserting Fastener Objects Using the Fastener Tool
Bolts

D Swing Eye Bolts

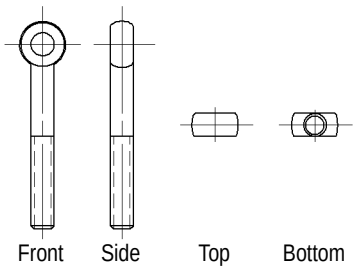
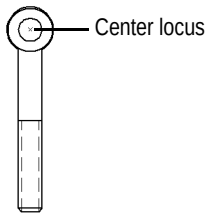
To insert a swing eye bolt:

1. Click the **Swing Eye Bolt** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Specify the swing eye bolt size in diameter/threads per inch (imperial) or diameter/thread pitch (metric)
Eye Inside Dia	Specify the inside diameter of the eye
Outside Diameter	Specify the outside diameter
Length	Specify the length under the bolt head
Thread Length	Specify the thread length
Thread Style (2D only)	Select the thread style; see "Fastener Object Thread Styles" on page 1397

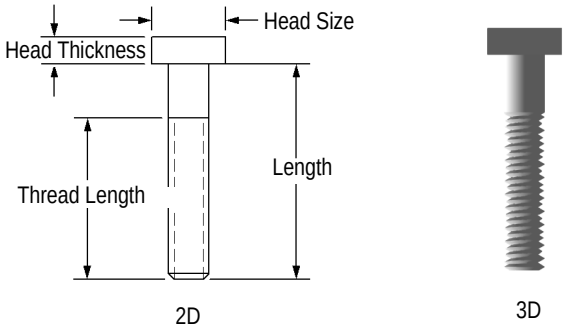
Parameter	Description
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the swing eye bolt with center line(s)
Show Threads (3D only)	Select to create the swing eye bolt with threads 
Place Locus at Center	Select to draw the swing eye bolt with a center locus 

Inserting Fastener Objects Using the Fastener Tool
Bolts

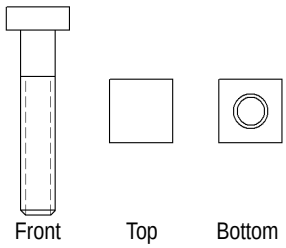
D T-bolts

To insert a T-bolt:

1. Click the **T-Bolt** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Specify the T-bolt size in diameter/threads per inch (imperial) or diameter/ thread pitch (metric)
Head Size	Specify the T-bolt head size
Head Thickness	Specify the T-bolt head thickness
Length	Specify the length under the bolt head
Thread Length	Specify the thread length
Thread Style (2D only)	Select the thread style; see “Fastener Object Thread Styles” on page 1397
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the T-bolt with center line(s)
Show Threads (3D only)	Select to create the T-bolt with threads 

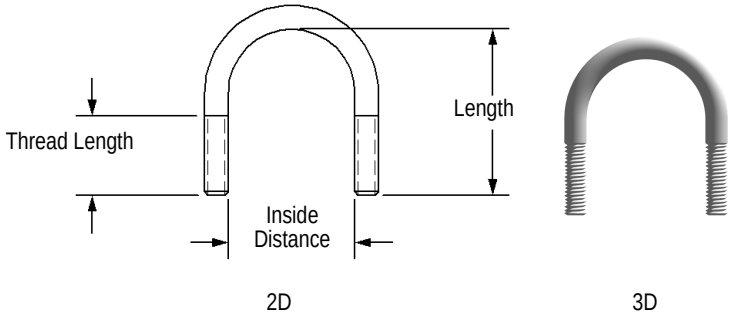
Inserting Fastener Objects Using the Fastener Tool
Bolts

U-bolts

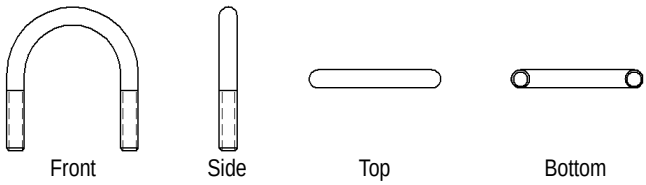
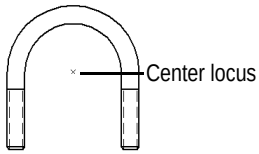
To insert a U-bolt:

1. Click the **U-Bolt** tool from the Fasteners tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Specify the U-bolt size in diameter/threads per inch (imperial) or diameter/thread pitch (metric)
Inside Distance	Specify the inside distance between the ends of the U-bolt
Length	Specify the length from the inner portion of the "U" to the end of the legs
Thread Length	Specify the thread length
Thread Style (2D only)	Select the thread style; see "Fastener Object Thread Styles" on page 1397
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the U-bolt with center line(s)
Place Locus at Center	Select to draw the U-bolt with a center locus 
Show Threads (3D only)	Select to create the U-bolt with threads 

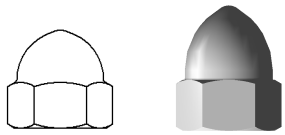
Inserting Fastener Objects Using the Fastener Tool
Bolts

D Nuts

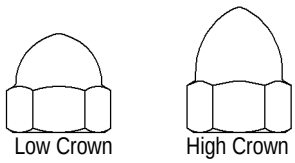
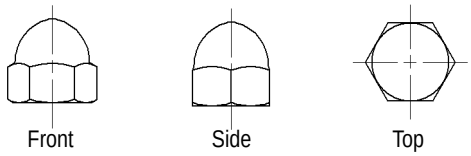
Acorn Nuts

To insert an acorn nut:

1. Click the **Acorn Nut** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

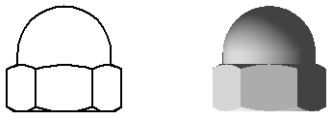
Parameter	Description
Size	Select the acorn nut size
Style	Specify the acorn nut style 
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the acorn nut with center line(s)
Show Hole (3D only)	Select to create the acorn nut with a center hole 

Inserting Fastener Objects Using the Fastener Tool
Hex Cap Nuts
Knurled Thumb Nuts
Wing Nuts

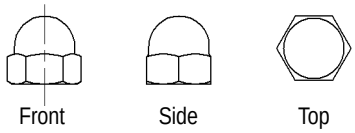
Hex Cap Nuts

To insert a hex cap nut:

1. Click the **Hex Cap Nut** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Select the hex cap nut size
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the hex cap nut with center line(s)
Show Center Hole (3D only)	Select to create the hex cap nut with a center hole 

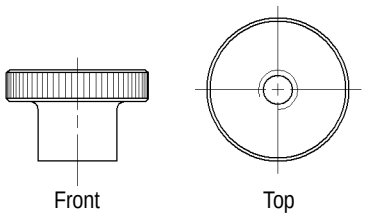
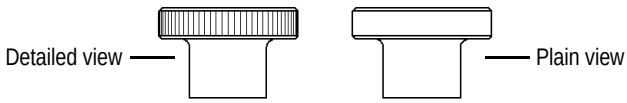
Inserting Fastener Objects Using the Fastener Tool
Acorn Nuts
Knurled Thumb Nuts
Wing Nuts

D Knurled Thumb Nuts

To insert a knurled thumb nut:

1. Click the **Knurled Thumb Nut** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Size	For knurled thumb nut (ASME inch), specify the size/threads per inch and style. For knurled thumb nut (DIN) specify the size/thread pitch.
View (2D only)	Select the 2D view 
Show Detail (2D only)	Indicate whether to display a detailed view of the knurled thumb nut 
Show Center Line (2D only)	Select to draw the knurled thumb nut with center line(s)

Inserting Fastener Objects Using the Fastener Tool
Acorn Nuts
Hex Cap Nuts
Wing Nuts

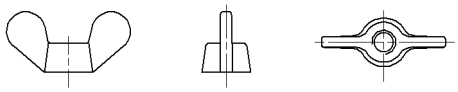
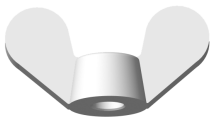
D Wing Nuts

To insert a wing nut:

1. Click the **Wing Nut (DIN)** tool from the Fasteners tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Size	Select the wing nut size
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the wing nut with center line(s)
Show Hole (3D only)	Select to create the wing nut with a center hole 

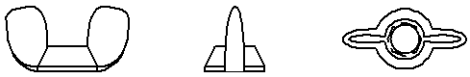
Wing Nuts - Type A
Wing Nuts - Type B
Wing Nuts - Type C
Wing Nuts - Type D

Wing Nuts - Type A

To insert a type A wing nut:

1. Click the **Wing Nut - Type A** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Size	Select the wing nut size
View (2D only)	Select the 2D view  Front Side Top
Show Center Line (2D only)	Select to draw the wing nut with center line(s)
Show Hole (3D only)	Select to create the wing nut with a center hole 

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Wing Nuts
Wing Nuts - Type B
Wing Nuts - Type C
Wing Nuts - Type D

Wing Nuts - Type B

To insert a type B wing nut:

1. Click the **Wing Nut - Type B** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

2D 3D

Click to show/hide the parameters.

Parameter	Description
Style	Select the wing nut style  Style 1 Style 2
Size	Select the wing nut size
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the wing nut with center line(s)
Show Hole (3D only)	Select to create the wing nut with a center hole 

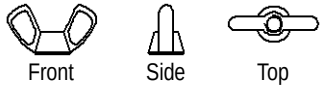
Wing Nuts
Wing Nuts - Type A
Wing Nuts - Type C
Wing Nuts - Type D

Wing Nuts - Type C

 To insert a type C wing nut:

1. Click the **Wing Nut - Type C** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Style	Select the wing nut style 
Size	Select the wing nut size
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the wing nut with center line(s)

Parameter	Description
Show Hole (3D only)	Select to create the wing nut with a center hole 

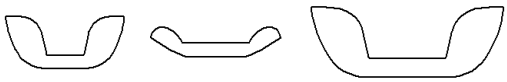
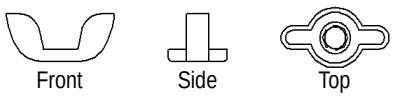
Wing Nuts
Wing Nuts - Type A
Wing Nuts - Type B
Wing Nuts - Type D

Wing Nuts - Type D

To insert a type D wing nut:

1. Click the **Wing Nut - Type D** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Style	Select the wing nut style 
Size	Select the wing nut size
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the wing nut with center line(s)
Show Hole (3D only)	Select to create the wing nut with a center hole 

Wing Nuts
Wing Nuts - Type A

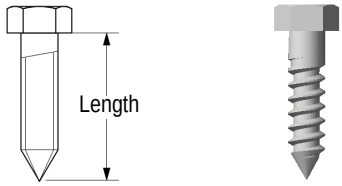
Wing Nuts - Type B
Wing Nuts - Type C

Screws

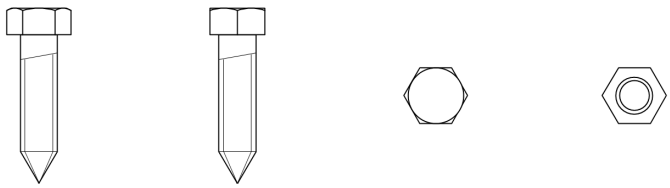
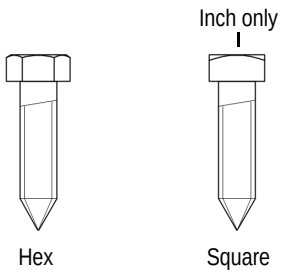
Lag Screws

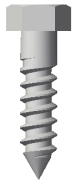
To insert a lag screw:

1. Click the **Lag Screw** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
View (2D only)	Select the 2D view 
Series	Select inch or metric to display the appropriate nominal sizes
Head Type	Select the lag screw head type 
Nominal Size	Select an inch or metric nominal size
Length	Enter the length under the screw head

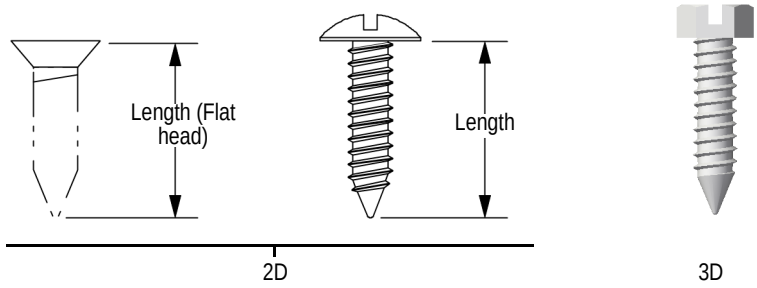
Parameter	Description
Show Threads	Select to create the lag screw with threads 
Show Center Line (2D only)	Select to draw the 2D lag screw with center line(s)

Inserting Fastener Objects Using the Fastener Tool
Sheet Metal Screws
Thumb Screws
Wood Screws

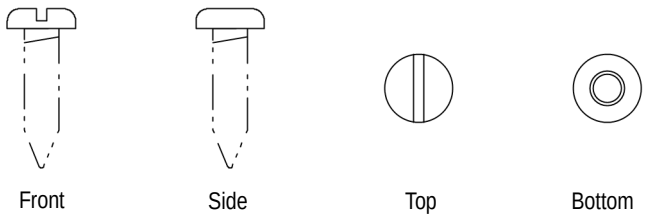
Sheet Metal Screws

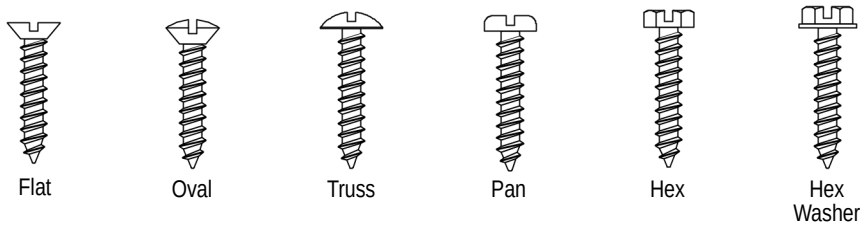
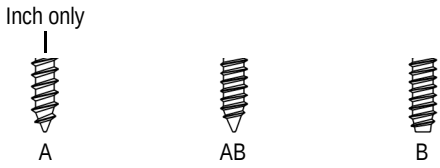
To insert a sheet metal screw (self-tapping screw):

1. Click the **Sheet Metal Screw** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
View (2D only)	Select the 2D view 

Parameter	Description
Series	Select inch or metric to display the appropriate nominal sizes
Head Type	Select the sheet metal screw head type 
Slot Type	Specify the slot type 
Point Type	Select the sheet metal screw point type 
Nominal Size	Select an inch or metric nominal size
Length	Enter the length under the screw head (or overall length for flat head screws)
Show Threads	Select to create the sheet metal screw with threads 
Show Center Line (2D only)	Select to draw the sheet metal screw with center line(s)

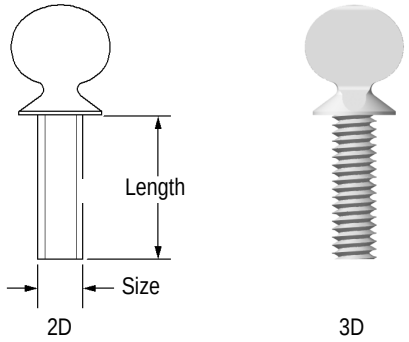
Inserting Fastener Objects Using the Fastener Tool
Lag Screws
Thumb Screws
Wood Screws

Thumb Screws

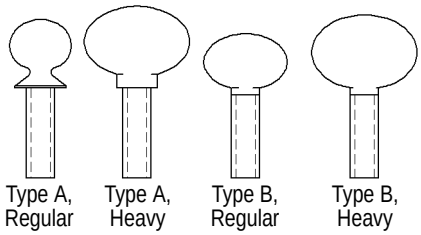
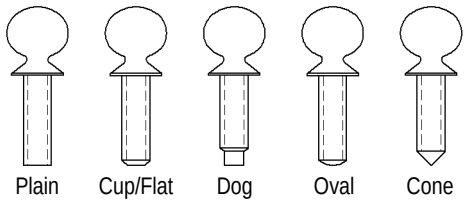
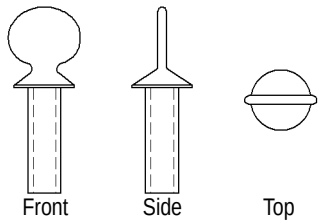
To insert a thumb screw:

1. Click the **Thumb Screw** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.

3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Screw Type	Specify the thumb screw type 
Point Type	Select the thumb screw point type 
Size	Select the thumb screw size
Length	For a custom length value, deselect **Use Standard Length Increments** and enter the length under the screw head
Thread Style (2D only)	Select the thread style; see “Fastener Object Thread Styles” on page 1397
View (2D only)	Select the 2D view 
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on size, per ASME standards; deselect to enter a custom length

Parameter	Description
Show Center Line (2D only)	Select to draw the thumb screw with center line(s)
Show Threads (3D only)	Select to create the 3D thumb screw with threads 

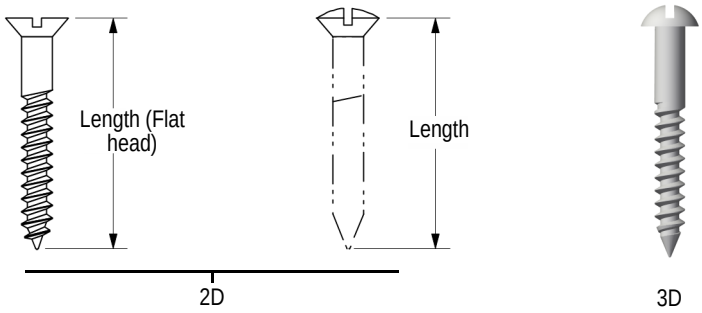
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting Fastener Objects Using the Fastener Tool
Lag Screws
Sheet Metal Screws
Wood Screws

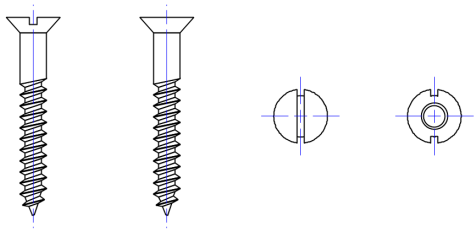
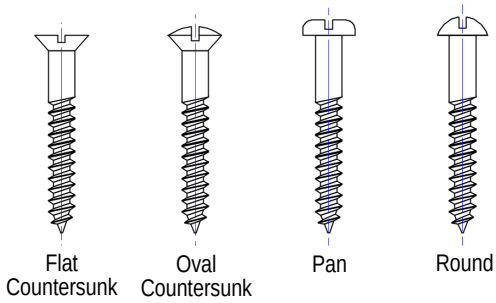
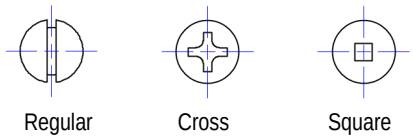
D Wood Screws

 To insert a wood screw:

1. Click the **Wood Screw** tool from the Detailing tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
View (2D only)	Select the 2D view 
Series	Select inch or metric to display the appropriate nominal sizes
Head Type	Select the wood screw head type 
Nominal Size	Select an inch or metric nominal size
Length	For a custom length value, deselect **Use Standard Length Increments** and enter the length under the screw head (or overall length for flat head screws)
Slot Type	Specify the slot type 
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on size; deselect to enter a custom length
Show Threads	Select to create the wood screw with threads
Show Center Line (2D only)	Select to draw the wood screw with center line(s)

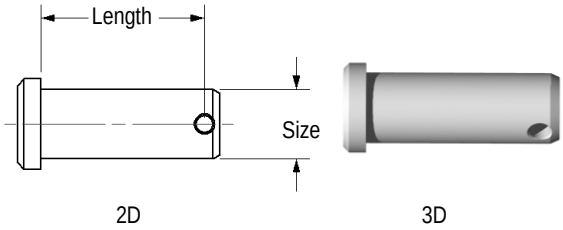
Inserting Fastener Objects Using the Fastener Tool
Lag Screws
Sheet Metal Screws
Thumb Screws

Pins

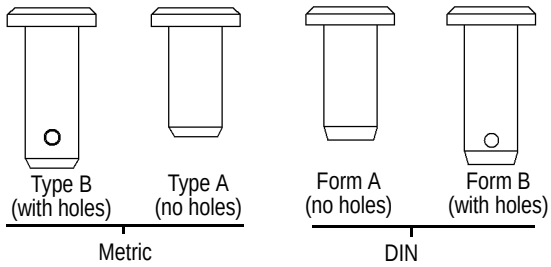
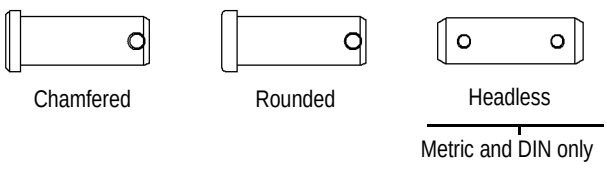
Clevis Pins

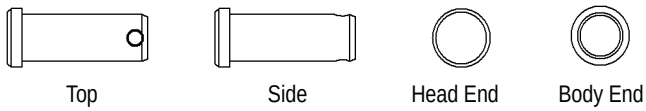
To insert a clevis pin:

1. Click the **Clevis Pin** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

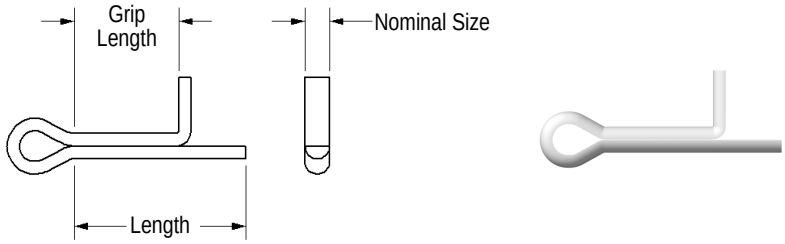
Parameter	Description
Size	Select the clevis pin size
Use Standard Length (inch only)	Select to automatically adjust the length to the recommended standard length per ASME standards; deselect to enter a custom length value
Pin Type (metric and DIN only)	Specify the pin type for metric and DIN clevis pins 
Length	For a custom length value, deselect **Use Standard Length** (inch) or **Use Standard Length Increments** (metric and DIN) and enter the desired length
Head Style	Select the head style 

Parameter	Description
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the clevis pin with center line(s)
Use Standard Length Increments (metric and DIN only)	Select to automatically adjust the length to the nearest increment based on standard sizes; deselect to enter a custom length

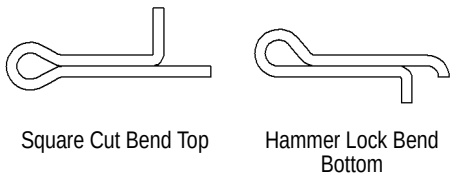
D Cotter Pins

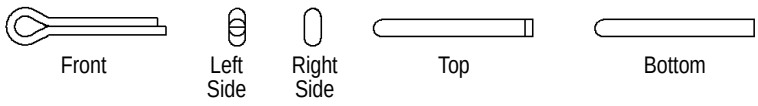
To insert a cotter pin:

1. Click the **Cotter Pin** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

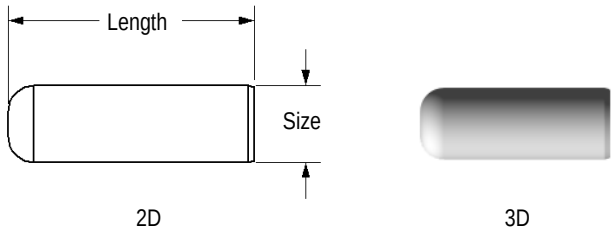
Parameter	Description
Cotter Pin Type	Select square cut or hammer lock 
Nominal Size	Select the nominal size in inches
Grip Length	Enter the grip length
Length	Enter the pin length
Bend Angle (Top)/ (Bottom)	Enter a bend angle in degrees for the top and/or bottom section of the cotter pin

Parameter	Description
View (2D only)	Select the 2D view 

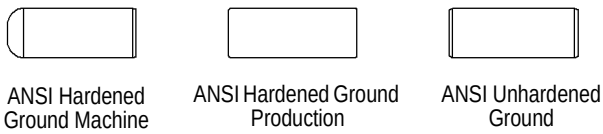
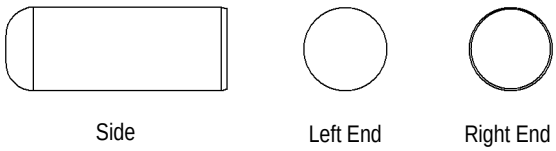
Dowel Pins

To insert a dowel pin:

1. Click the **Dowel Pin** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



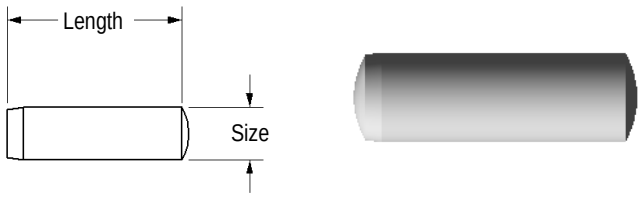
Click to show/hide the parameters.

Parameter	Description
Dowel Pin Type	Select the type of ANSI dowel pin 
Nominal Size	Select a nominal size in inches
Length	Enter the pin length
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the dowel pin with center line(s)

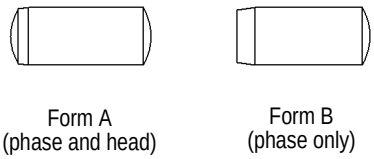
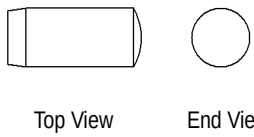
D Parallel Pins

To insert a parallel pin:

1. Click the **Parallel Pin** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

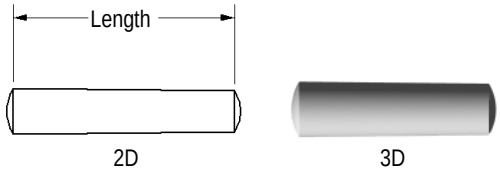
Parameter	Description
Size	Select the parallel pin size
Pin Type	Select the pin type 
Length	For a custom length, deselect **Use Standard Length Increments** and enter the length value
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the parallel pin with center line(s)
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on DIN standard sizes; deselect to enter a custom length

D Taper Pins

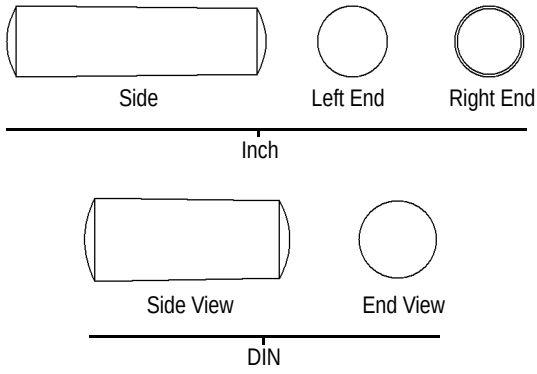
To insert a taper pin:

1. Click the **Taper Pin** tool from the Fasteners tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Select a taper pin size
Length	For a custom length value, deselect **Use Standard Length Range** (inch) or **Use Standard Length Increments** (DIN) and enter the desired length
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the taper pin with center line(s)
Use Standard Length Range (inch only)	Select to automatically constrain the length to the standard range per ASME specifications; deselect to enter a length value outside of the range
Use Standard Length Range and Increments (DIN only)	Select to automatically constrain the length to the standard range per DIN specifications; deselect to enter a length value outside of the range

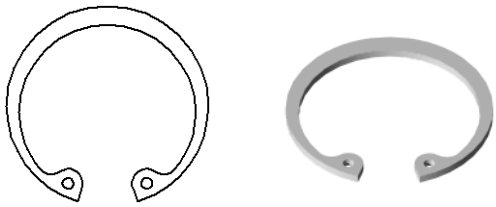
D Retaining Rings and Washers

Retaining Rings

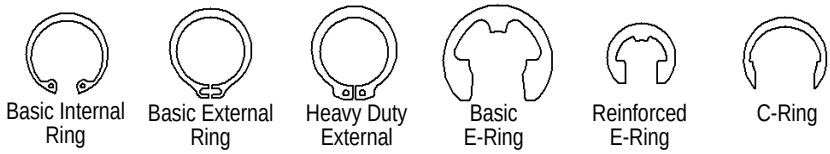
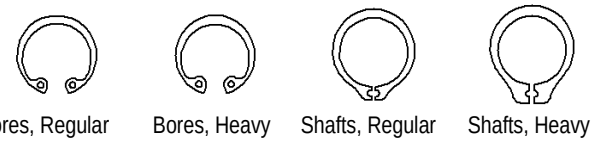
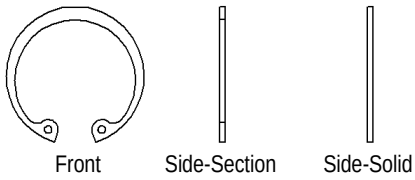
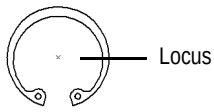
 To insert a retaining ring:

1. Click the **Retaining Ring** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.

3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Type	Select the retaining ring type
ASME	
DIN	
Size (mm)	Select the retaining ring size in millimeters
View (2D only)	Select the 2D view 
Place Locus at Center	Select to add a locus at the center of the retaining ring 

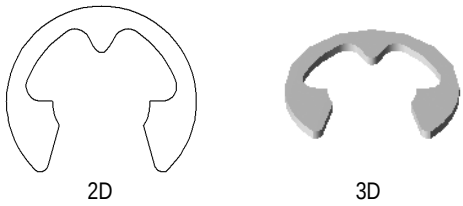
Retaining Washers

Retaining Washers

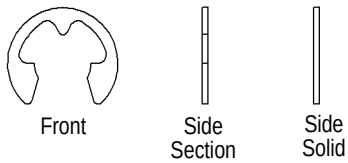
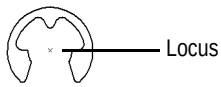
To insert a retaining washer:

1. Click the **Retaining Washer** tool from the Fasteners tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Groove Size (mm)	Select the retaining washer groove size in millimeters
View (2D only)	Select the 2D view 
Place Locus at Center	Select to add a locus at the center of the retaining washer 

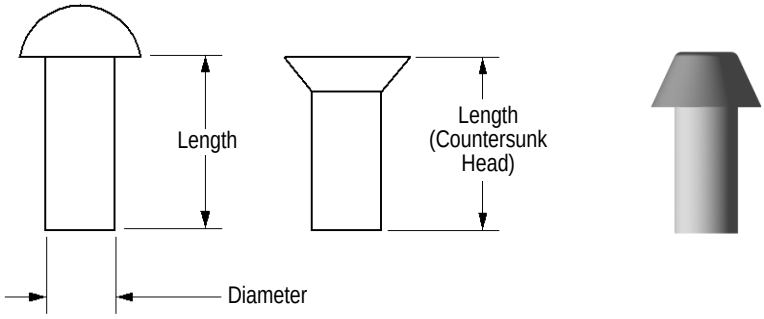
Retaining Rings

Rivets

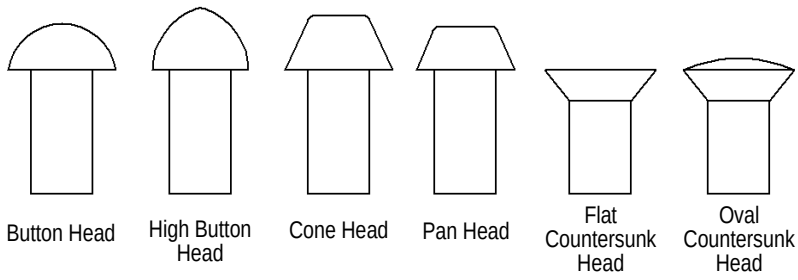
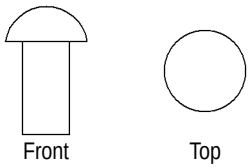
Large Rivets

To insert a large rivet:

1. Click the **Large Rivet** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Diameter	Select a diameter in inches
Length	Enter the length under the rivet head (or overall length for countersunk rivets)
Head Type #1	Select a head type for one end of the rivet 
Head Type #2	Select a head type, if any, for the other end of the rivet
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the rivet with center line(s)

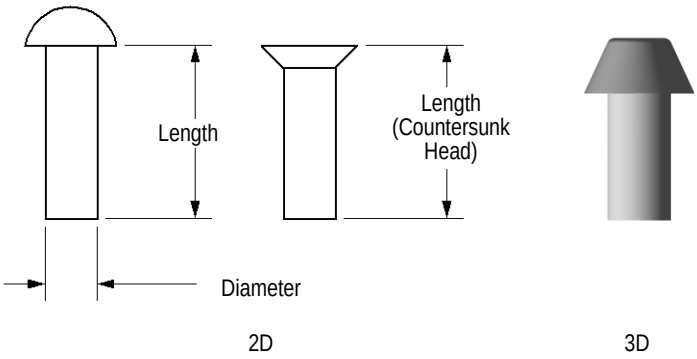
Small Rivets
Rivets (DIN)
Tubular Rivets

Small Rivets

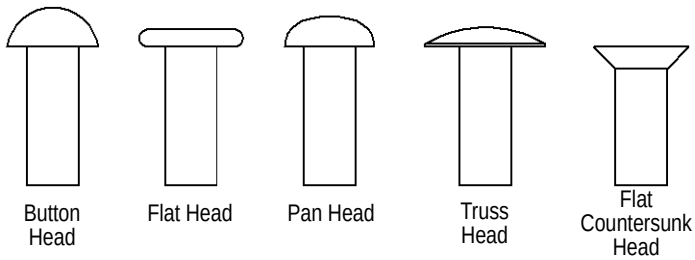
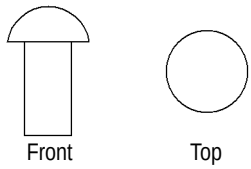
To insert a small rivet:

1. Click the **Small Rivet** tool from the Fasteners tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

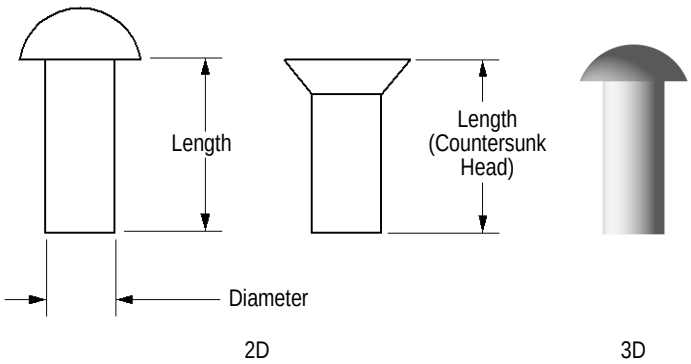
Parameter	Description
Diameter	Select a diameter in inches
Length	Enter the length under the rivet head (or overall length for the countersunk rivet)
Head Type #1	Select a head type for one end of the rivet 
Head Type #2	Select a head type, if any, for the other end of the rivet
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the rivet with center line(s)

Large Rivets
Rivets (DIN)
Tubular Rivets

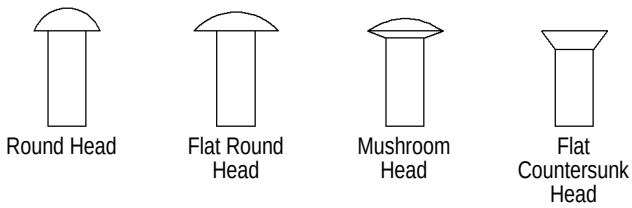
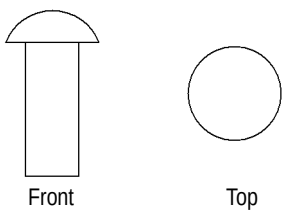
D Rivets (DIN)

To insert a rivet (DIN):

1. Click the **Rivet** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Diameter	Select a diameter in millimeters
Length	Enter the length under the rivet head (or overall length for the countersunk rivet)
Head Type #1	Select a head type for one end of the rivet 
Head Type #2	Select a head type, if any, for the other end of the rivet
View (2D only)	Select the 2D view 
Show Center Line (2D only)	Select to draw the rivet with center line(s)

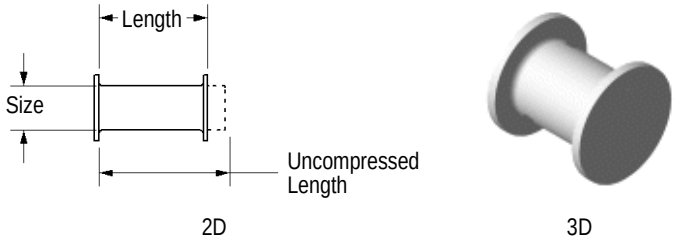
Large Rivets

Small Rivets
Tubular Rivets

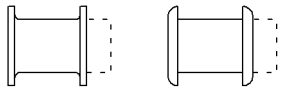
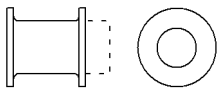
D Tubular Rivets

To insert a tubular rivet:

1. Click the **Tubular Rivet** tool from the Fasteners tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Size	Select a size in millimeters
Length	For a custom length value, deselect **Use Standard Length Increments** and enter the length
Form	Select one of the rivet types 
View (2D only)	Select the 2D view 
Show Uncompressed Length (2D only)	Select to display the uncompressed rivet length
Use Standard Length Increments	Select to automatically adjust the length to the nearest increment based on DIN standard sizes; deselect to enter a custom length value
Show Center Line (2D only)	Select to draw the rivet with center line(s)

Large Rivets
Small Rivets

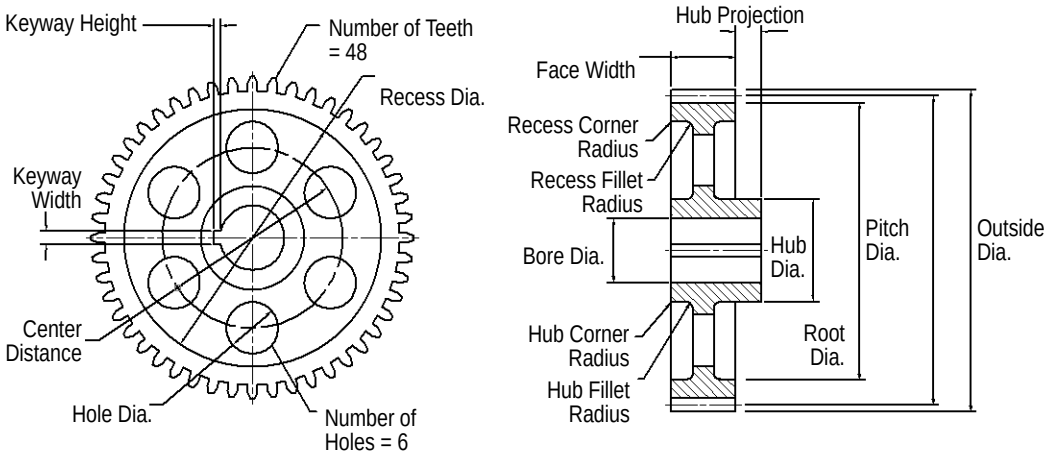
Rivets (DIN)

D Gears

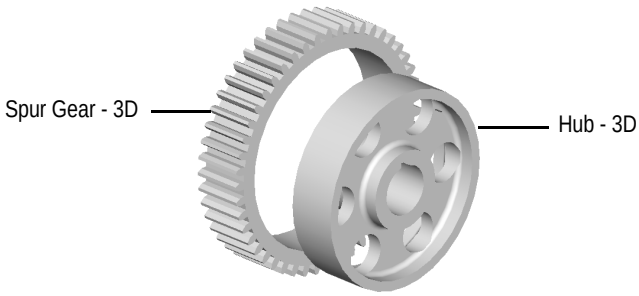
Spur Gears

To insert a spur gear:

1. Click the **Spur Gear** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

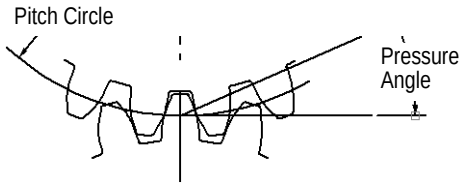
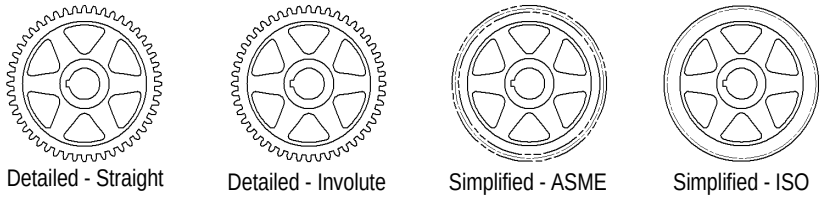


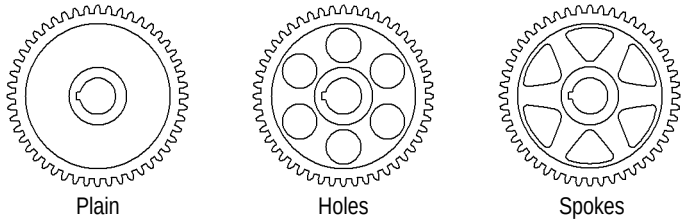
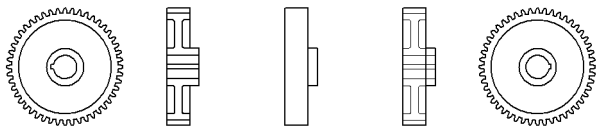
Spur Gear - 2D



Click to show/hide the parameters.

Parameter	Description
Pitch Diameter	Specify the pitch diameter
Number of Teeth	Enter the number of teeth

Parameter	Description
Pressure Angle	Enter the pressure angle 
Diametrical Pitch (Ref.)	Displays the diametrical pitch (for reference only)
Module (mm) (Ref.)	Displays the module (for reference only)
Outside Dia. (Ref.)	Displays the outside diameter (for reference only)
Root Dia. (Ref.)	Displays the root diameter (for reference only)
Tooth Profile	Select the type of tooth profile 
Face Width	Enter the width of the gear face
Draw Recess (2D only)	To draw a recessed web, select **Draw Recess** and specify the 2D parameters
Web Thickness	Enter the web thickness
Recess Diameter	Enter the recess diameter
Recess Corner Radius	Specify the recess corner radius
Recess Fillet Radius	Indicate the recess fillet radius
Draw Hub (2D only)	Select to include a hub and then specify the 2D parameters
Hub Diameter	Enter the hub diameter
Hub Projection (Left / Right)	Specify the amount of projection for the hub on both the left and the right; a negative value indicates that the hub face is recessed
Hub Corner Radius	Enter the hub corner radius
Hub Fillet Radius	Enter the hub fillet radius
Draw Bore (2D only)	Select to draw a bore and then specify the 2D parameters
Bore Diameter	Enter the bore diameter
Keyway	If a keyway is present, select the square, rectangular, or custom size; otherwise, select **None**; the square and rectangular selections apply the ASME-recommended size based on the bore diameter

Parameter	Description
Width/Height	For custom keyway sizes, enter the width and height values of the keyway
Web Configuration (2D only)	Select the web configuration and then specify the 2D parameters, if any. When **Holes** are selected, their size can be specified as a percentage or diameter value.  Plain Holes Spokes
Number	Indicate the number of holes or spokes for the gear (does not apply to **Plain** web configurations)
Size (10 - 100%)	For **Holes (Percent)** and **Spokes** web configurations, enter the percentage of the recess opening occupied by the holes or spokes
Center Distance	When **Holes** is selected for the web configuration, specify the distance between the hole centers
Hole Diameter	When **Holes** is selected for the web configuration, indicate the size of the holes
View (2D only)	Select the 2D view  Front Side - Section Side - Solid Side - Solid with Hidden Lines Back
Show Center Lines (2D only)	Select to draw the 2D gear with center lines
Helix Angle (3D only)	Indicate the 3D spur gear helix angle
Hole Diameter (3D only)	Indicate the hole diameter for the 3D spur gear; when including a hub 3D object in the drawing, adjust the spur gear hole diameter and hub outside diameter to fit together

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

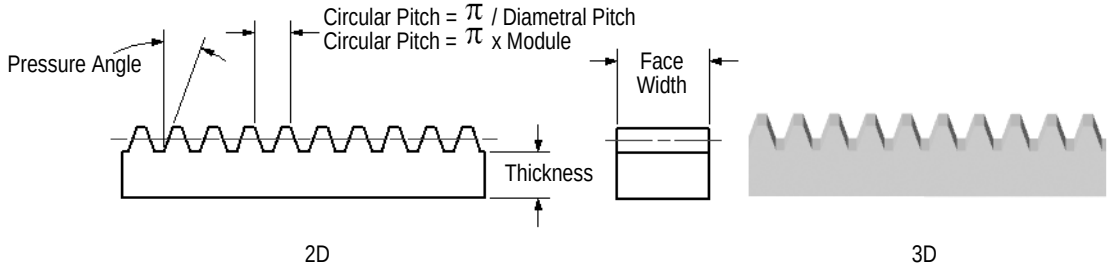
Spur Gear Racks
Bevel Gears
Worm Gear Sets

Spur Gear Racks

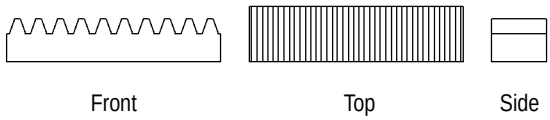
To insert a spur gear rack:

1. Click the **Spur Gear Rack** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Method	Select whether to insert the spur gear rack using either a diametral pitch (for imperial sizes) or module (for metric sizes)
Diametral Pitch or Module (mm)	Specify either the diametral pitch or module, for the selected insertion method
Pressure Angle	Select the pressure angle
Number of Teeth	Enter the number of teeth
Thickness	Enter the base thickness
Face Width	Enter the width of the spur gear rack face
View (2D only)	Select the 2D view 
Show All Teeth	Select to draw the all teeth or just the end teeth
Draw Fillet	Select to draw filleted spur gear rack teeth
Show Pitch Line (2D only)	Select to draw the spur gear rack with a pitch line

Spur Gears
Bevel Gears
Worm Gear Sets
Worm Gears

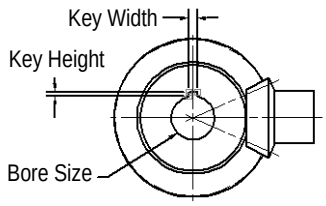
Bevel Gears

A bevel gear set consists of a gear and pinion.

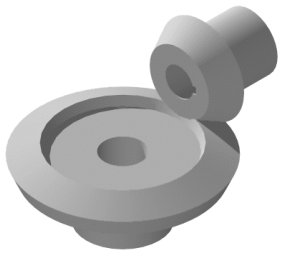
To insert a bevel gear:

1. Click the **Bevel Gears** tool from the Machine Components tool set.

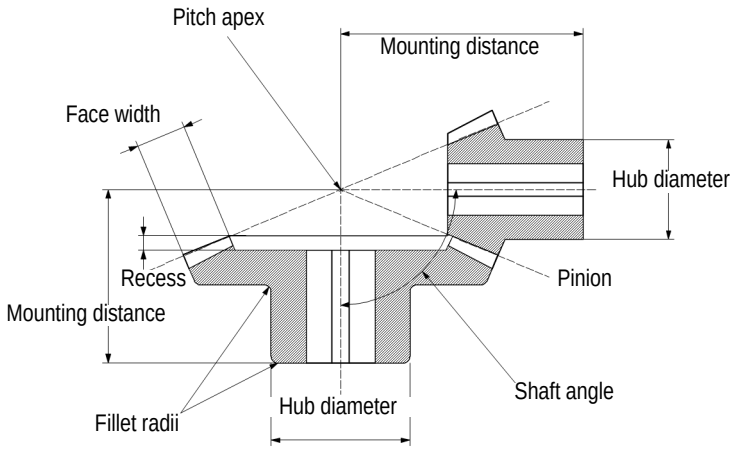
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



2D

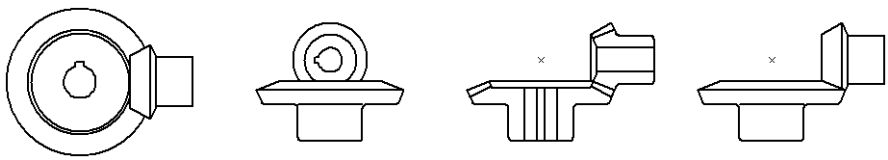


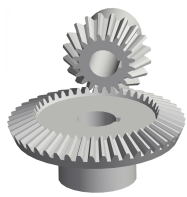
3D



2D

Click to show/hide the parameters.

Parameter	Description
View (2D only)	Select the 2D view 
Value to Use	Select either **Diametral Pitch** or **Module**, and then enter the relevant parameter
Diametral Pitch	Enter the diametral pitch
Module (mm)	Enter the module
Shaft Angle (deg.)	Enter the shaft angle in degrees
Face Width	Enter the face width
Configuration	Select to use the gear parameter values for only the gear, only the pinion, or for both the gear and the pinion; this can also be used to create miter gears

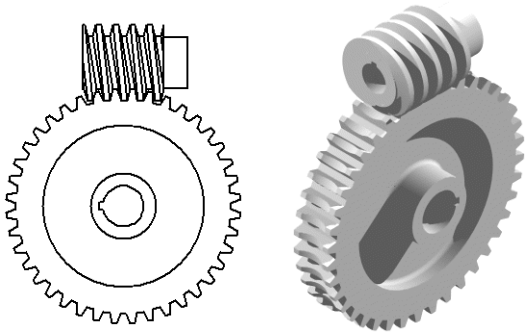
Parameter	Description
Gear Properties	Specify the parameters to draw the gear
No. of Teeth	Enter the number of teeth
Pitch Dia. (Ref.)	Displays the pitch diameter based on the diametral pitch and number of teeth
Mounting Distance	Enter the distance from the back of the gear to the pitch apex
Hub Diameter	Enter the hub diameter
Hub Projection	Enter the hub projection, for styles with a projected hub
Bore Diameter	Enter the bore diameter
Key Type	If the gear includes a keyway, select keyway shape (with ASME-recommended sizes), or select **Custom size**
Kwy Width/Height	For custom keyway sizes, enter the keyway width and height
Recess	If the gear has a recess, enter the recess value
Fillet Radius	Enter the fillet radius of the hub
Pinion Properties	For pinion only or gear and pinion configurations, specify the pinion parameters
No. of Teeth	Enter the number of teeth
Pitch Dia. (Ref.)	Displays the pitch diameter based on the diametral pitch and number of teeth
Mounting Distance	Enter the distance from the back of the pinion to the pitch apex
Hub Diameter	Enter the hub diameter
Hub Projection	Enter the hub projection, for styles with a projected hub
Bore Diameter	Enter the bore diameter
Key Type	If the pinion includes a keyway, select keyway shape (with ASME-recommended sizes), or select **Custom size**
Kwy Width/Height	For custom keyway sizes, enter the keyway width and height
Recess	If the pinion has a recess, enter the recess value
Fillet Radius	Enter the fillet radius of the hub
Show Pinion at Left (2D only)	Select to display the pinion on the left side of the bevel gear
Draw Center Lines (2D only)	Select to draw the bevel gear with center lines
Show Teeth (3D only)	Select to draw the bevel gear with teeth 
Spiral Angle (Deg.) (3D only)	To draw a 3D spiral bevel gear, specify the spiral angle degrees for the teeth

Spur Gears

Spur Gear Racks
Worm Gear Sets

Worm Gear Sets

Worm gear sets consist of a worm and worm gear. Create a worm gear set by following the instructions for creating both a worm and worm gear.



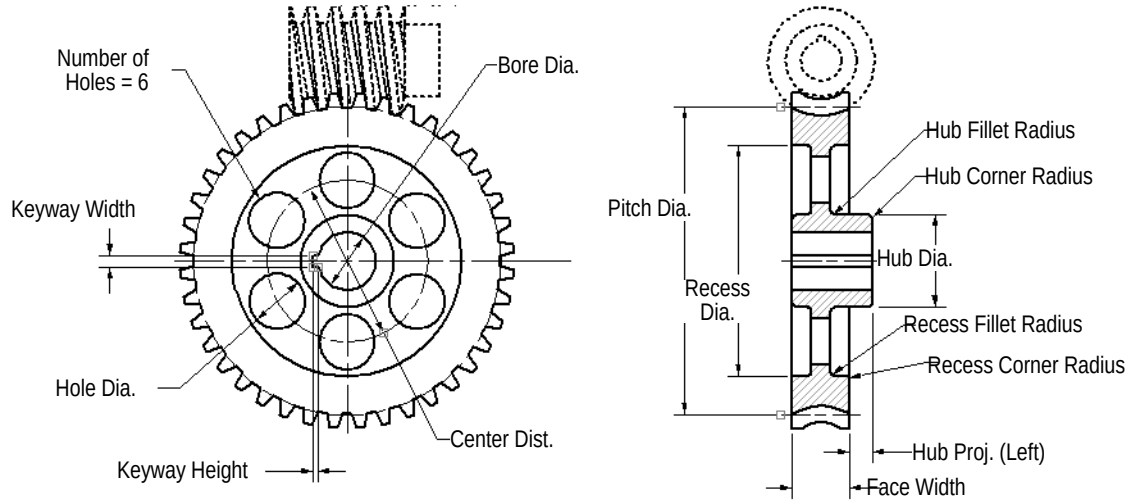
2D 3D

Worm Gears
Worms
Spur Gears
Spur Gear Racks
Bevel Gears

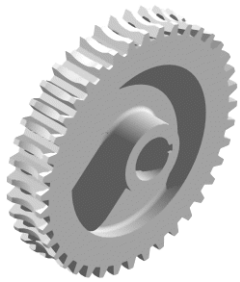
Worm Gears

To insert a worm gear:

1. Click the **Worm Gear** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

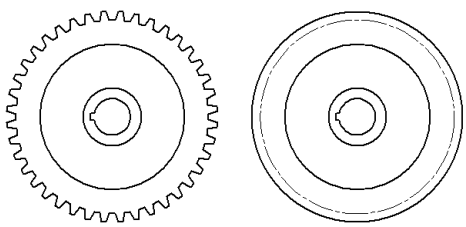


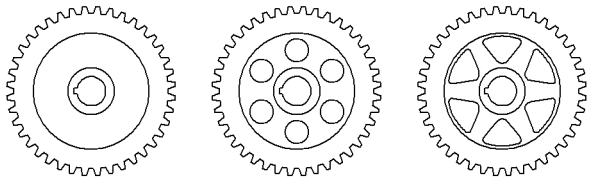
2D

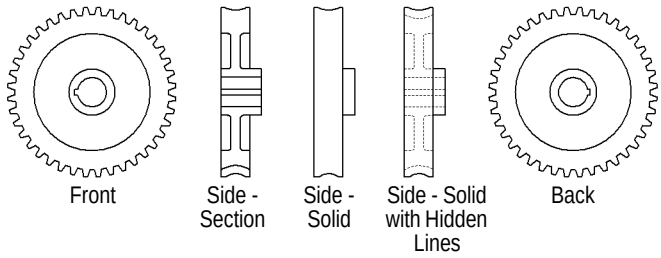


3D

Click to show/hide the parameters.

Parameter	Description
Circular Pitch	Enter the circular pitch
Number of Teeth	Enter the number of teeth
Pitch Diameter (Ref.)	Displays the pitch diameter of the gear, based on the pitch and number of teeth
Pressure Angle (deg.)	Select the pressure angle in degrees
Number of Starts (Worm)	Select the number of starts of the worm
Pitch Dia (Worm)	Specify the worm pitch diameter
Tooth Profile	Select the type of tooth profile  Detailed Schematic
Face Width	Enter the width of the gear face
Draw Recess (2D only)	Select **Draw Recess** to draw a recessed web, and specify the 2D parameters

Parameter	Description
Web Thickness	Enter the web thickness
Recess Diameter	Enter the recess diameter
Recess Corner Radius	Specify the recess corner radius
Recess Fillet Radius	Indicate the recess fillet radius
Draw Hub (2D only)	Select to include a hub and then specify the 2D parameters
Hub Diameter	Enter the hub diameter
Hub Projection (Left / Right)	Specify the amount of projection for the hub on both the left and the right; a negative value indicates that the hub face is recessed
Hub Corner Radius	Enter the hub corner radius
Hub Fillet Radius	Enter the hub fillet radius
Draw Bore (2D only)	Select to draw a bore and then specify the 2D parameters
Bore Diameter (2D and 3D)	Enter the bore diameter
Keyway	If a keyway is present, select the square, rectangular, or custom size; otherwise, select **None**. The square and rectangular selections apply the ASME-recommended size based on the bore diameter.
Width/Height	For custom keyway sizes, enter the width and height values of the keyway
Web Configuration (2D only)	Select the gear configuration and then specify the 2D parameters, if any. When **Holes** are selected, their size can be specified as a percentage or diameter value.  Plain Holes Spokes
Number	Indicate the number of holes or spokes for the gear (does not apply to **Plain** web configurations)
Size (10 - 100%)	For **Holes (Percent)** and **Spokes** web configurations, enter the percentage of the recess opening occupied by the holes or spokes
Center Distance	When **Holes** is selected for the web configuration, specify the distance between the hole centers
Hole Diameter	When **Holes** is selected for the web configuration, indicate the size of the holes

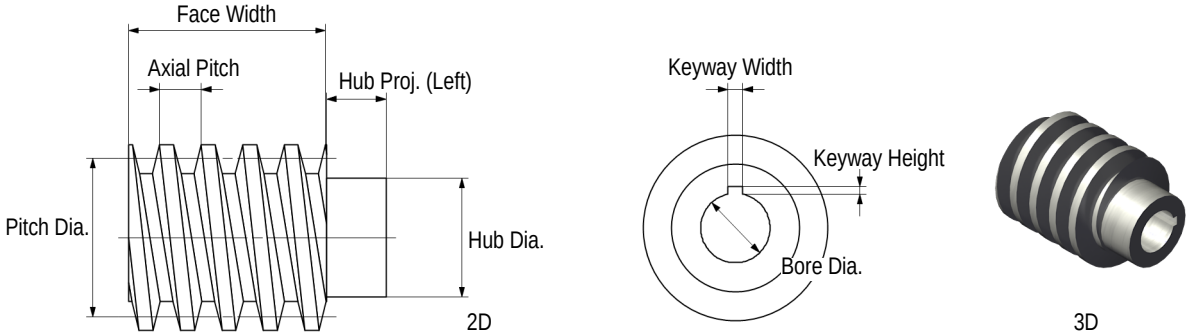
Parameter	Description
View (2D only)	Select the 2D view 
Show Center Lines (2D only)	Select to draw the gear with center lines

Worms

Worms

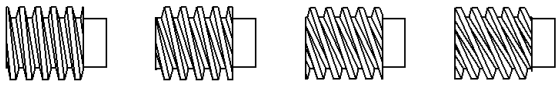
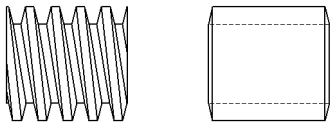
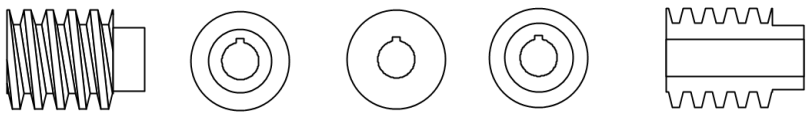
To insert a worm:

1. Click the **Worm** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Pitch Diameter	Enter the pitch diameter
Axial Pitch	Enter the axial pitch
Pressure Angle (deg.)	Select the pressure angle in degrees

Parameter	Description
Number of Starts	Select the number of starts  Single Double Triple Quad
Tooth Profile	Select the type of tooth profile  Detailed Schematic
Face Width	Enter the width of the worm face
Draw Hub(s)	Select to include a hub and then specify the parameters
Hub Diameter	Enter the hub diameter
Hub Projection (Left/Right)	Specify the amount of projection for the hub on both the left and the right; a negative value indicates that the hub face is recessed
Draw Bore	Select to draw a bore and then specify the parameters
Bore Diameter	Enter the bore diameter
Keyway	If a keyway is present, select the square, rectangular, or custom size; otherwise, select **None**. The square and rectangular selections apply the ASME-recommended size based on the bore diameter.
Keyway Width/Height	For custom keyway sizes, enter the width and height values of the keyway
View (2D only)	Select the 2D view  Front Right Side Left Side Side - Section Front - Section
Show Center Lines (2D only)	Select to draw the worm with center lines

Worm Gears

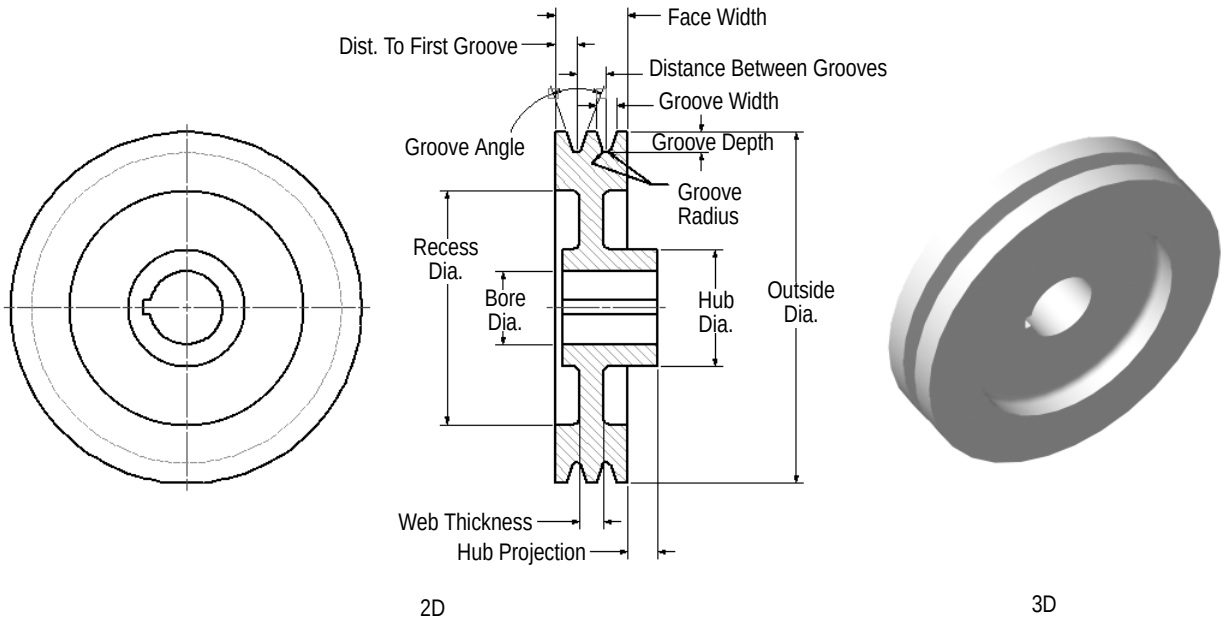
Pulleys

The pulley object accommodates both round belts and V-belts, and up to ten grooves.

To insert a pulley:

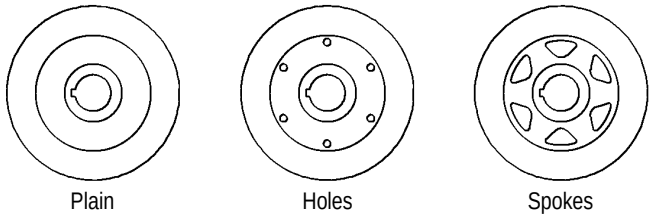
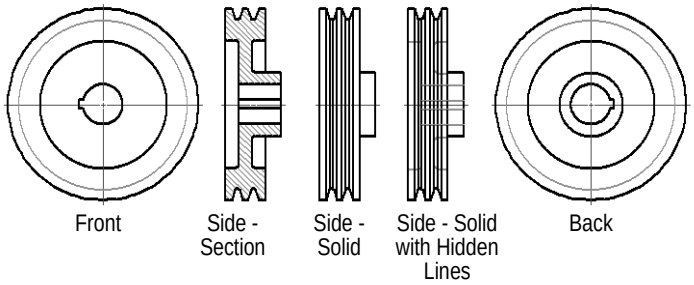
1. Click the **Pulley** tool from the Machine Components tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

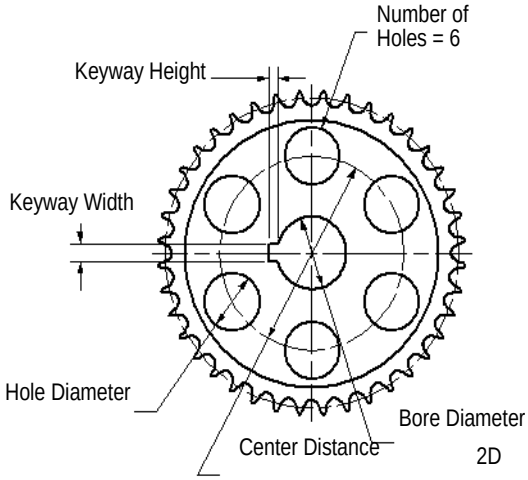
Parameter	Description
Outside Diameter	Enter the outside diameter
Groove Depth	Enter the groove depth
Groove Width	Enter the groove width
Groove Angle (deg.)	Enter the groove angle
Groove Radius	Enter the groove radius
Number of Grooves	Select the number of grooves (up to ten)
Dist. to First Groove	For multiple-groove pulleys, enter the distance from the left edge of the pulley to the center of the first groove
Distance Between Grooves	For multiple-groove pulleys, enter the distance between grooves
Face Width	For multiple-groove pulleys, this value is automatically calculated; for single-groove pulleys, enter the face width
Draw Recess (2D only)	Select **Draw Recess** to draw a recessed web, and specify the 2D parameters
Web Thickness	Enter the web thickness
Recess Diameter	Enter the recess diameter
Recess Corner Radius	Specify the recess corner radius
Recess Fillet Radius	Specify the recess fillet radius
Draw Hub (2D only)	Select to include a hub and then specify the 2D parameters
Hub Diameter	Enter the hub diameter

Parameter	Description
Hub Projection (Left / Right)	Specify the amount of projection for the hub on both the left and the right; a negative value indicates that the hub face is recessed
Hub Corner Radius	Enter the hub corner radius
Hub Fillet Radius	Enter the hub fillet radius
Draw Bore (2D only)	Select to draw a bore and then specify the 2D parameters
Bore Diameter	Enter the bore diameter
Keyway	If a keyway is present, select the square, rectangular, or custom size; otherwise, select **None**. The square and rectangular selections apply the ASME-recommended size based on the bore diameter.
Width/Height	For custom keyway sizes, enter the width and height values of the keyway
Web Configuration (2D only)	Select the web configuration and then specify the 2D parameters, if any. When **Holes** are selected, their size can be specified as a percentage or diameter value.  Plain Holes Spokes
Number	Indicate the number of holes or spokes for the sprocket (does not apply to **Plain** web configurations)
Size (10 - 100%)	For **Holes (Percent)** and **Spokes** web configurations, enter the percentage of the recess opening occupied by the holes or spokes
Center Distance	For **Holes** web configuration, specify the distance between the hole centers
Hole Diameter (2D and 3D)	For **Holes** web configuration, indicate the size of the holes
View (2D only)	Select the 2D view  Front Side - Section Side - Solid Side - Solid with Hidden Lines Back
Show Center Lines (2D only)	Select to draw the pulley with center lines
Show Root Diameter (2D only)	Select to draw the pulley with a root diameter

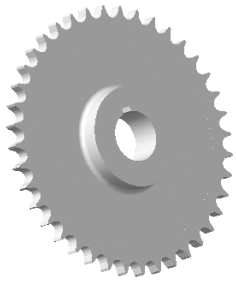
Sprockets

To insert a sprocket:

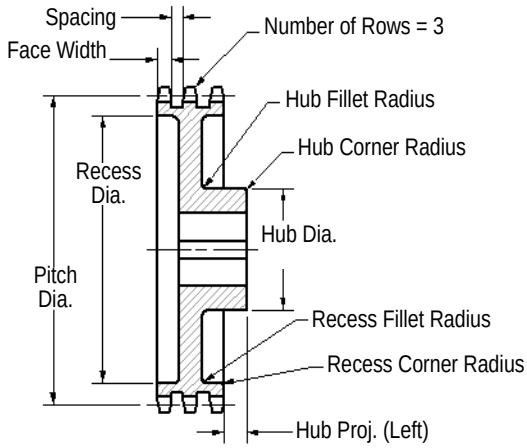
1. Click the **Sprocket** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



2D

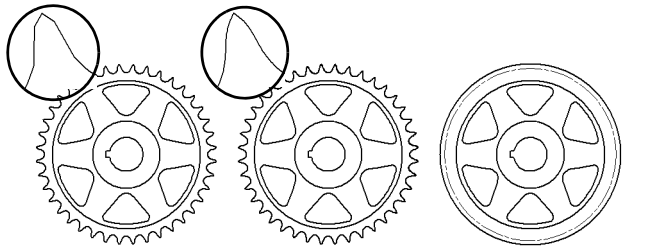
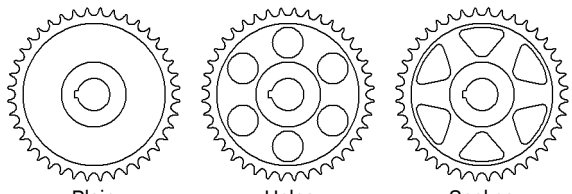


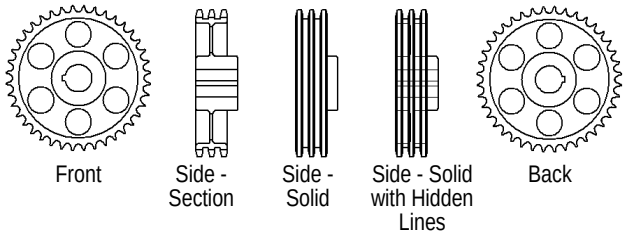
3D



Click to show/hide the parameters.

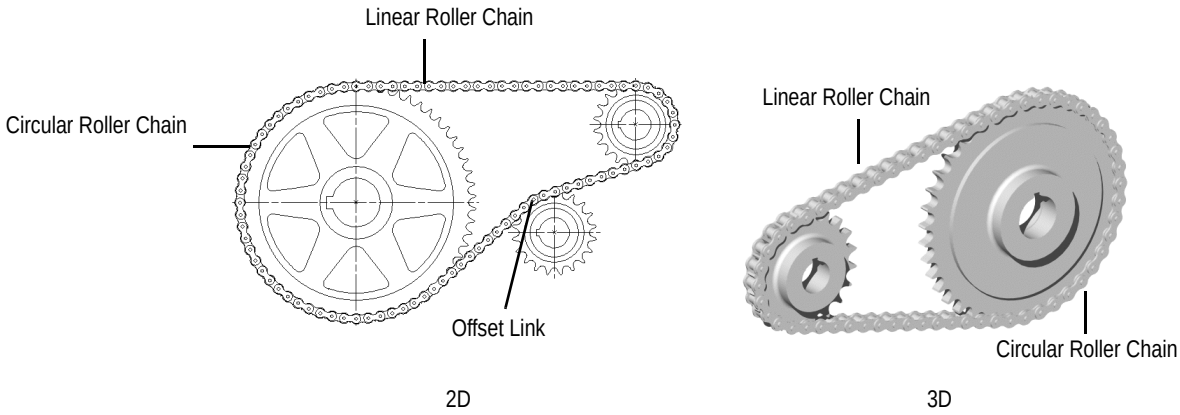
Parameter	Description
Std. Chain No. (Pitch)	Select the standard ANSI or ISO roller chain number; the pitch is given as reference
Number of Teeth	Enter the number of teeth
Pitch Diameter (Ref.)	Displays the pitch diameter based on the pitch and number of teeth
Number of Rows	Enter the number of rows of teeth

Parameter	Description
Tooth Profile	Select the type of tooth profile 
Use Standard Width and Spacing	Select to use standard **Face Width** and **Spacing** values based on the selected **Pitch**; deselect to specify custom values and enter the values
Face Width	Enter the face width value
Spacing	Enter the spacing between rows of teeth
Draw Recess (2D only)	Select **Draw Recess** to draw a recessed web, and specify the 2D parameters
Web Thickness	Enter the web thickness
Recess Diameter	Enter the recess diameter
Recess Corner Radius	Specify the recess corner radius
Recess Fillet Radius	Specify the recess fillet radius
Draw Hub (2D only)	Select to include a hub and then specify the 2D parameters
Hub Diameter	Enter the hub diameter
Hub Projection (Left / Right)	Specify the amount of projection for the hub on both the left and the right; a negative value indicates that the hub face is recessed
Hub Corner Radius	Enter the hub corner radius
Hub Fillet Radius	Enter the hub fillet radius
Draw Bore (2D only)	Select to draw a bore and then specify the 2D parameters
Bore Diameter	Enter the bore diameter
Keyway	If a keyway is present, select the square, rectangular, or custom size; otherwise, select **None**. The square and rectangular selections apply the ASME-recommended size based on the bore diameter.
Width/Height	For custom keyway sizes, enter the width and height values of the keyway
Web Configuration (2D only)	Select the web configuration and then specify the 2D parameters, if any. When **Holes** are selected, their size can be specified as a percentage or diameter value. 
Number	Indicate the number of holes or spokes for the sprocket (does not apply to **Plain** web configurations)

Parameter	Description
Size (10 - 100%)	For **Holes (Percent)** and **Spokes** web configurations, enter the percentage of the recess opening occupied by the holes or spokes
Center Distance	When **Holes** is selected for the web configuration, specify the distance between the hole centers
Hole Diameter (2D and 3D)	When **Holes** is selected for the web configuration, indicate the size of the holes
View (2D only)	Select the 2D view 
Draw Center Lines (2D only)	Select to draw the sprocket with center lines

Roller Chains

To create a complete roller chain, it is usually necessary to combine linear and circular sections. Offset links may be required to connect chain sections.



2D 3D

Linear Roller Chains
Circular Roller Chains
Offset Links

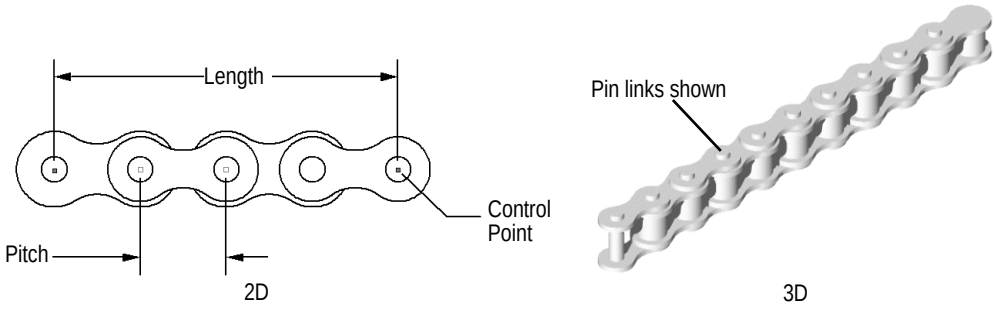
Linear Roller Chains

The 2D linear roller chain is a linear object, while the 3D linear roller chain is a point object.

To insert a linear roller chain:

1. Click the **Roller Chain - Linear** tool from the Machine Components tool set.

2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

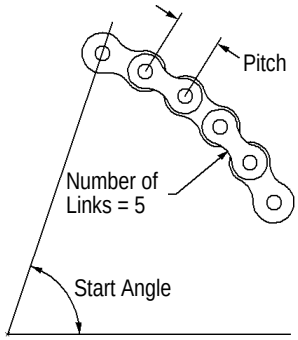
Parameter	Description
Std. Chain No. (Pitch)	Select the standard ANSI or ISO roller chain number; the pitch is given as a reference
Length (2D only)	Specify the length of the 2D roller chain by entering a value here or by clicking and dragging one of the chain control points; as the length increases, the number of links increases (depending on the selected pitch)
No. of Links	Enter the number of links; the length automatically adjusts
Start with Pin Link	Select to begin the chain section with a pin link; otherwise, the section begins with a roller link
Show Pins (3D only)	Select to show the pins

Circular Roller Chains
Offset Links

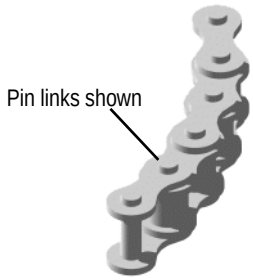
D Circular Roller Chains

To insert a circular roller chain:

1. Click the **Roller Chain - Circular** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

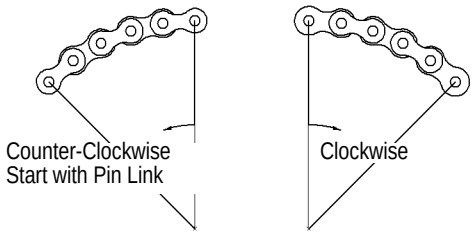
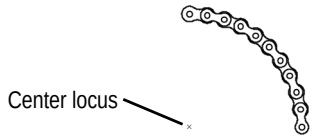


2D



3D

Click to show/hide the parameters.

Parameter	Description
Standard Chain No. (Pitch)	Select the standard ANSI or ISO roller chain number; the pitch is given as a reference
No. of Teeth (Sprocket)	Enter the number of teeth on the associated sprocket
Pitch Diameter (Ref.)	Displays the pitch diameter, based on the pitch value and the number of teeth
Start Angle	Specify the angle where the chain starts
Number of Links	Enter the number of links in the chain; these will be added or subtracted from the end of the chain
Direction	Select **Clockwise** or **Counterclockwise** 
Start with Pin Link	Select to begin the chain section with a pin link; otherwise, the section begins with a roller link
Show Pins (3D only)	Select to show the pins
Place Locus at Center	Select to draw the roller chain with a center locus 

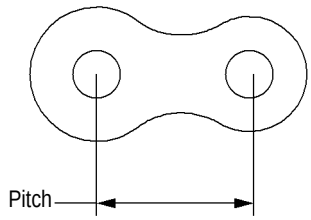
Linear Roller Chains
Offset Links

D Offset Links

The offset link connects a roller link and a pin link.

To insert an offset link:

1. Click the **Roller Chain - Offset Link** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Standard Chain No. (Pitch)	Select the standard ANSI or ISO roller chain number; the pitch is given as a reference
Show Pins (3D only)	Select to show the pins

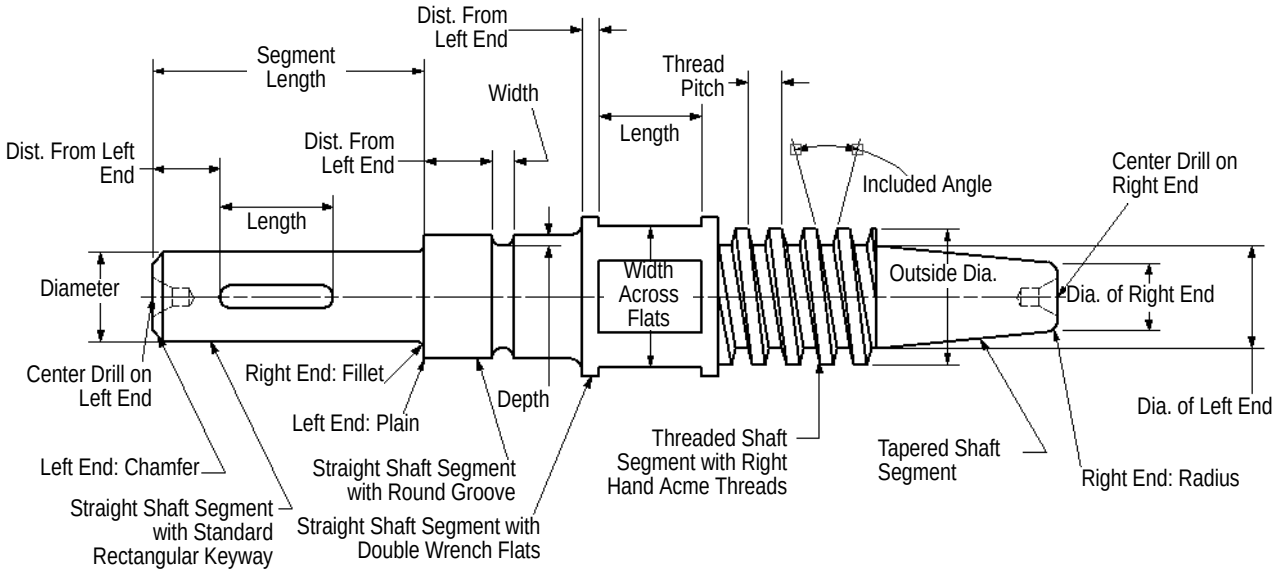
Linear Roller Chains
Circular Roller Chains

Shafts

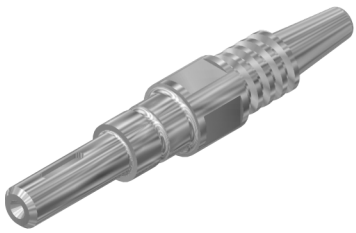
The **Shaft** tool allows the creation of a shaft consisting of multiple straight, tapered, and threaded shaft segments.

To insert a shaft segment:

1. Click the **Shaft** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.
4. Click the **Configuration** button in the Object Info palette to add, delete, or modify shaft component segments (see "Shaft Segment Properties" on page 1455).



2D shaft with five segments



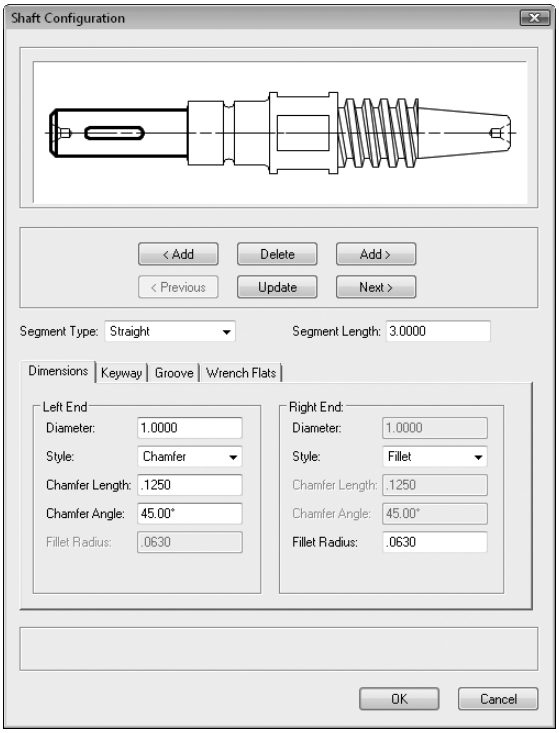
3D shaft with five segments

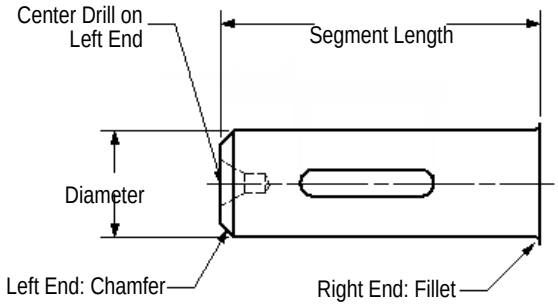
Shaft Segment Properties

Shaft Segment Properties

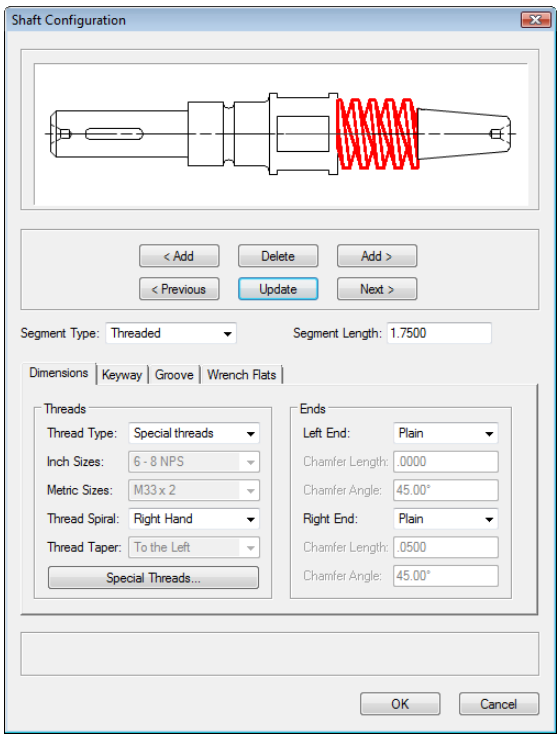
To add, delete, or modify component segments to construct the desired shaft object:

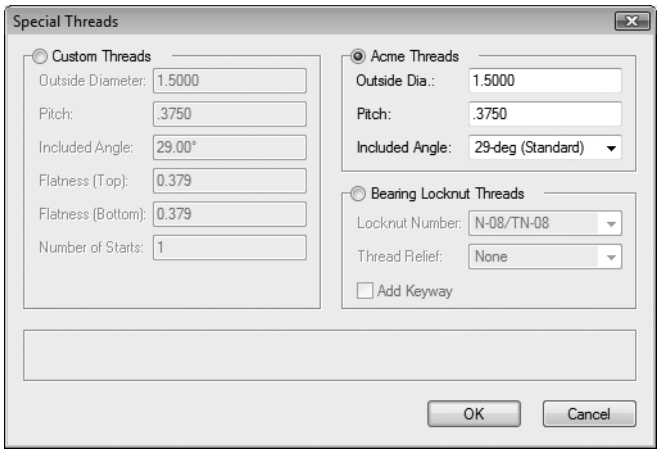
1. Select a shaft in the drawing file and click the **Configuration** button in the Object Info palette.
2. While the currently selected segment is highlighted, specify the shaft segment parameters on each tab to define the shaft. Click the **Update** button to dynamically change the preview when changes are made.
3. Click the Dimensions tab to specify the configuration of the shaft segment.

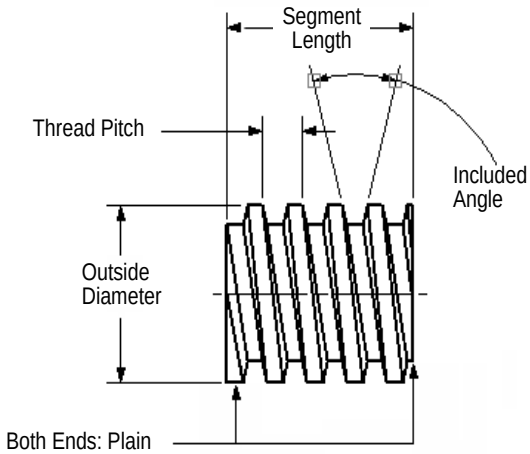




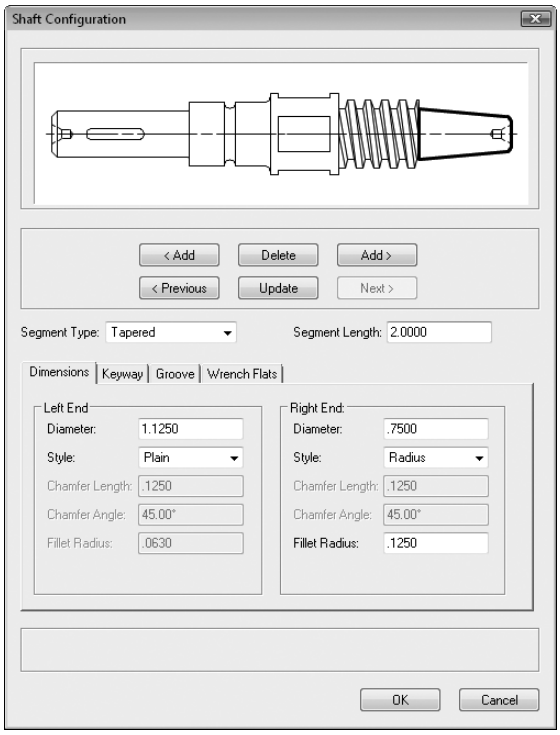
Straight Shaft Segment with Standard Rectangular Keyway

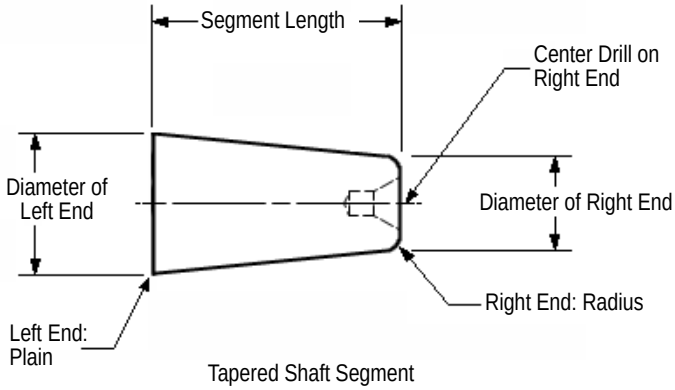






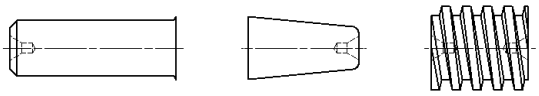
Threaded Shaft Segment with Right Hand Acme Threads





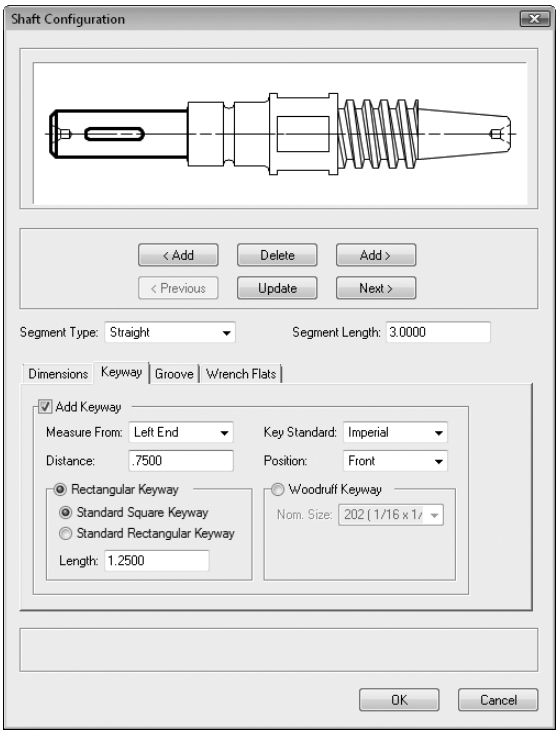
Tapered Shaft Segment

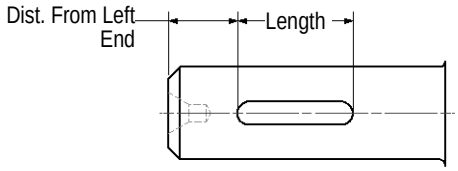
Click to show/hide the parameters.

Parameter	Description
< Add / Add >	Click **< Add** to add a shaft segment to the left of the current segment or click **Add >** to add a shaft segment to the right of the current segment
Delete	Deletes the current shaft segment
< Previous / Next >	Click **Previous** to select the shaft segment to the left of the current segment or click **Next** to select the shaft segment to the right of the current segment
Update	Updates the preview image for the current segment
Segment Type	Select a straight, tapered, or threaded shaft segment type  Straight Tapered Threaded
Segment Length	Specify the shaft segment length
Diameter	For straight or tapered segments, enter the diameter of the ends
Style	For straight or tapered segments, select the end style
Chamfer Length	For straight or tapered segments, specify the length for chamfer/flare ends styles
Chamfer Angle	For straight or tapered segments, specify the angle value for chamfer/flare ends styles
Fillet Radius	For straight or tapered segments, specify the radius value for fillet/radius end styles

Parameter	Description
Thread Type	For threaded segments, select the standard inch or metric thread series, or select special threads to specify custom, Acme, or bearing locknut thread parameters; the selection in this field enables the associated thread size parameter or **Special Threads** button
Inch Sizes	For standard inch series, select the thread size and threads per inch
Metric Sizes	For standard metric series, select the thread size and pitch
Thread Spiral	For threaded segments, select whether to draw left hand or right hand threads
Thread Taper	For tapered pipe threads, select whether the taper points to the left or right
Special Threads	For threaded segments, click **Special Threads** to specify custom, acme, or bearing locknut thread parameters. • For custom threads, enter the outside diameter of threads, the thread pitch, the included angle of the threads, the flatness at both the top (at the outside diameter) and bottom (at the root diameter) of the threads, and the number of starts. • For Acme threads, enter the outside diameter of threads, the thread pitch, and select the included angle of the threads. • For bearing locknut threads, select the bearing locknut number, the thread relief position (if any), and select to add a keyway, if desired. Click **OK** to return to the Shaft Configuration dialog box.
Left End / Right End	For threaded segments, select plain or chamfered for the left and right ends
Chamfer Length	For threaded segments, specify the length for chamfer end styles
Chamfer Angle	For threaded segments, specify the angle value for chamfer end styles

4. Click the Keyway tab to specify the configuration of the shaft segment keyway (straight **Segment Type** only).



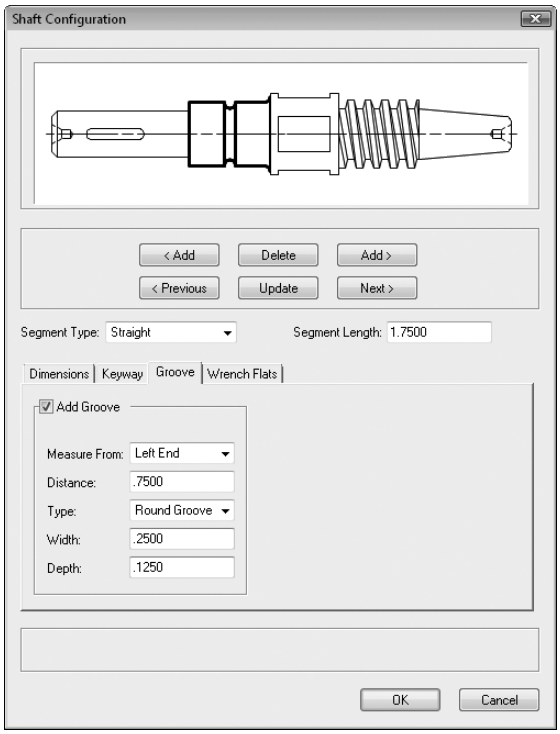


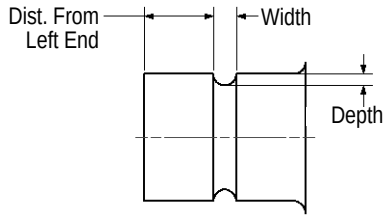
Straight Shaft Segment with Rectangular Keyway

Click to show/hide the parameters.

Parameter	Description
Add Keyway	For straight segments, select **Add Keyway** to add a keyway to the current segment
Measure From	Select to measure the position of the keyway from the left or right end
Distance	Enter the distance from the end of the segment to the keyway
Key Standard	Select to insert either an imperial or metric keyway
Position	Select to position the keyway in the front, top, bottom, or back of the segment
Parallel Keyway/ Rectangular Keyway	Select to add a rectangular or parallel keyway; specify either a standard square or rectangular keyway, and enter the keyway length
Woodruff Keyway	Select to add a woodruff keyway and select the nominal size

5. Click the Groove tab to specify the configuration of the shaft segment groove (straight **Segment Type** only).



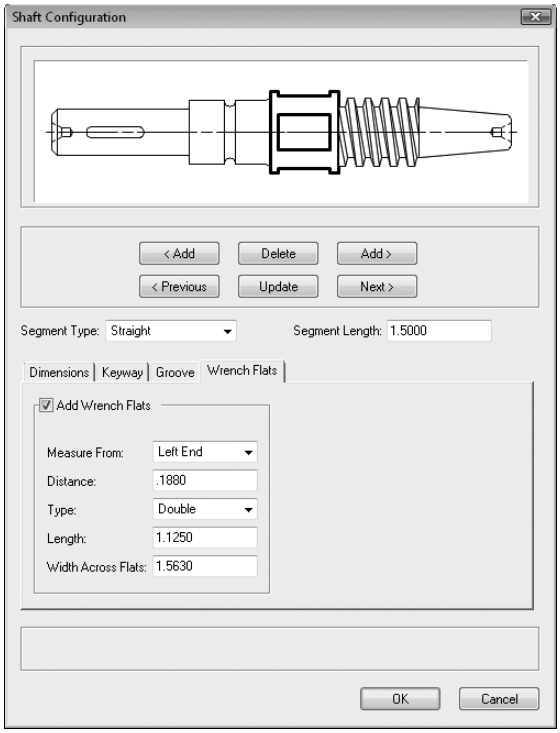


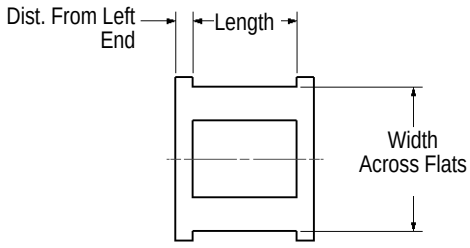
Straight Shaft Segment with Round Groove

Click to show/hide the parameters.

Parameter	Description
Add Groove	For straight segments, select **Add Groove** to add a groove to the current segment
Measure From	Select to measure the position of the groove from the left or right end
Distance	Enter the distance from the end of the segment to the groove
Type	Select to add a round, rectangular, or V-groove
Width	Enter the groove width
Depth	Enter the groove depth

6. Click the Wrench Flats tab to specify the configuration of the shaft segment wrench flats (straight **Segment Type** only).





Straight Shaft Segment with Double Wrench Flats

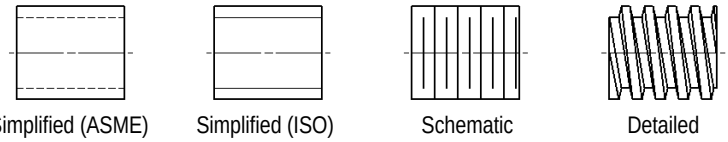
Click to show/hide the parameters.

Parameter	Description
Add Wrench Flats	For straight segments, select **Add Wrench Flats** to add wrench flats to the current segment
Measure From	Select to measure the position of the wrench flats from the left or right end
Distance	Enter the distance from the end of the segment to the groove
Type	Select to add single or double wrench flats
Length	Enter the wrench flats length
Width Across Flats	Enter the width across flats of the wrench flats

7. Click **OK** to set the shaft parameters and close the Shaft dialog box.
8. To edit parameters, click **Configuration** in the Object Info palette, double-click the shaft, or select **Edit** from the context menu to open the Shaft dialog box. The following additional parameters can be modified directly in the Object info palette.

Click to show/hide the parameters.

Parameter	Description
Configuration	Click **Configuration** to open the Shaft Configuration dialog box
Add Center Drill	Select to draw the shaft with a center drill. Select either a plain or bell configuration for the center drill, select the size, and whether a center drill is drawn on the left, right, or both ends of the shaft.

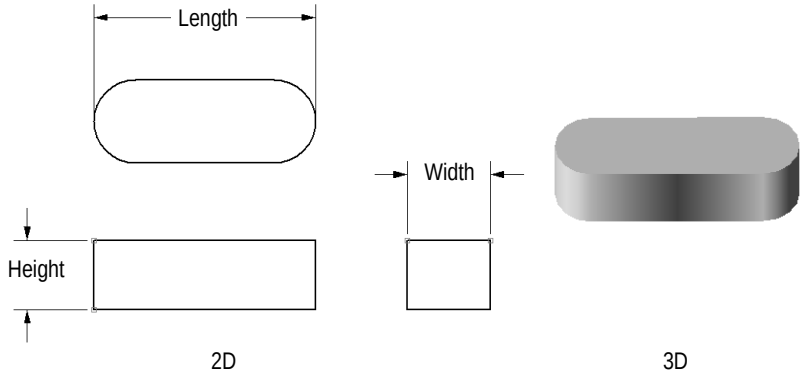
Parameter	Description
Show Threads As	Select how to draw the 2D threaded shaft threads  Simplified (ASME) Simplified (ISO) Schematic Detailed
Show Center Line	Select to draw the 2D shaft with center line(s)
Show Threads	Select to draw the 3D shaft with threads
Create Solid	Select to create a generic solid from the individual 3D segments. For large, complex shafts, this will greatly reduce file size. Deselect this option when editing the shaft to speed the editing process.

Shafts
Shaft Break 2
Shaft Break

Keys

To insert a key:

1. Click the **Key** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Standard	Select Imperial or Metric
Nom. Size (in) or (mm)	Select one of the standard key sizes
Length	Enter the key length

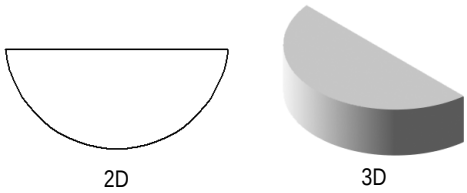
Parameter	Description
View (2D only)	Select the 2D view  Top Front Side

Woodruff Keys
Keyways

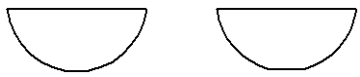
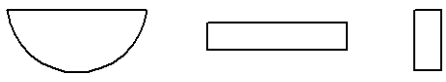
Woodruff Keys

To insert a woodruff key:

1. Click the **Woodruff Key** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Standard	Select Imperial or Metric
Nom. Size (in) or (mm)	Select one of the standard key sizes
Key Type	Select the key type  Full Radius Flat Bottom
Depth of Keyseat (shaft)/Depth of Keyseat (hub)	Displays the depth of the keyseat shaft and hub, determined by the standard size selected
View (2D only)	Select the 2D view  Front Top Side

Keys

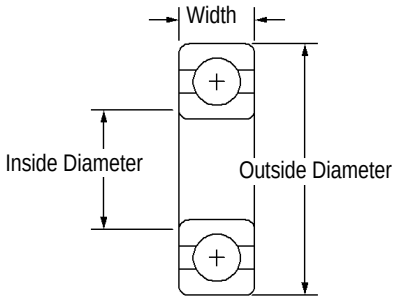
Keyways

Bearings

Ball Bearings

To insert a ball bearing:

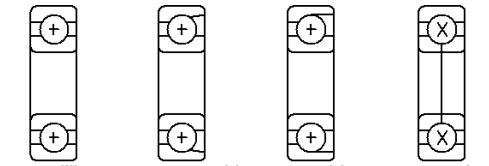
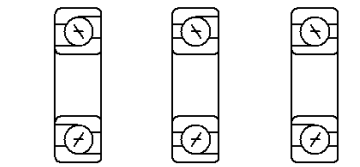
1. Click the **Ball Bearing** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

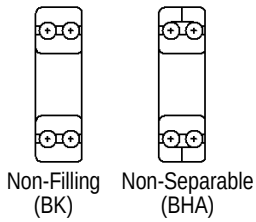
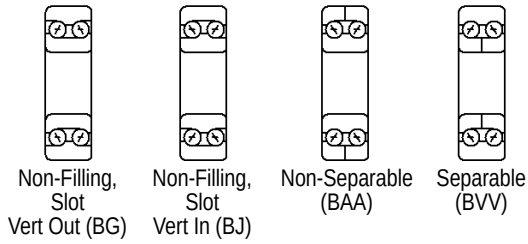
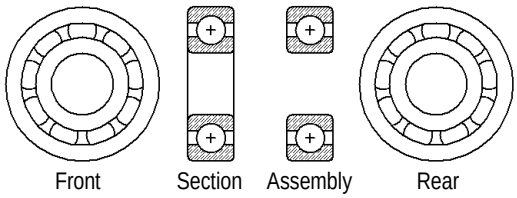
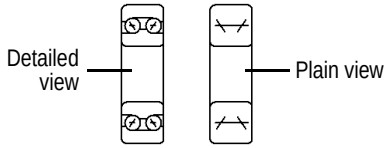


2D

3D
(Section with rollers highlighted in black)

Click to show/hide the parameters.

Parameter	Description
Bearing Type	Select from a variety of single ball or double ball radial and angular bearings
Single Row, Radial	 Non-Filling Slot (BC) Non-Separable (BH) Separable (BM) Four Point
Single Row, Angular	 Non-Separable (BN) Separable Outer Ring (BNS) Separable Inner Ring (BAS)

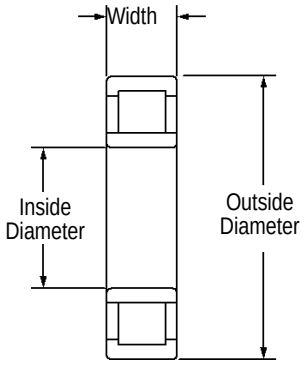
Parameter	Description
Double Row, Radial	
Double Row, Angular	
Inside Diameter	Specify the inside diameter of the bearing
Outside Diameter	Specify the outside diameter of the bearing
Width	Enter the width of the bearing
View (2D only)	Select the 2D view 
Show Detail (2D only)	Indicate whether to display a detailed 2D view of the bearing 
Show Center Line (2D only)	Select to draw the bearing with center lines
Section (3D only)	Select to display a section of the bearing
Highlight Rollers (3D only)	Select to highlight the rollers within the bearing in black

D Roller Bearings

To insert a roller bearing:

1. Click the **Roller Bearing** tool from the Machine Components tool set.

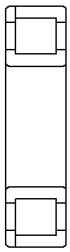
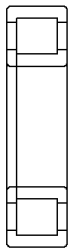
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

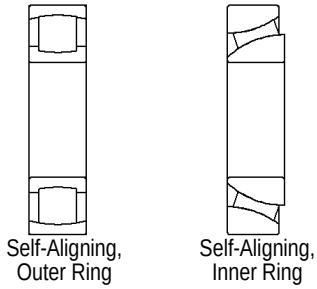
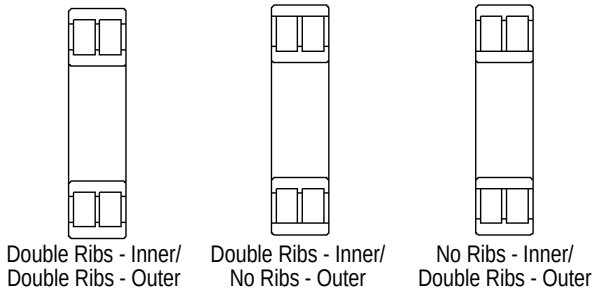
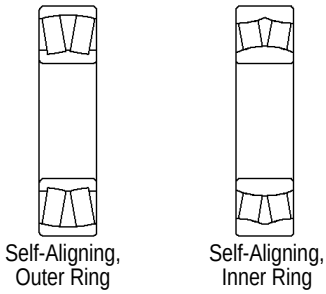
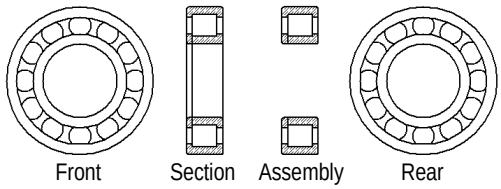
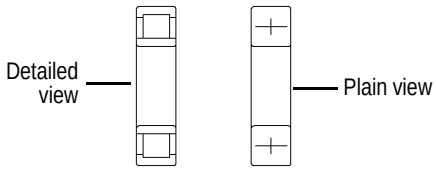


2D

3D
(Section with rollers highlighted in black)

Click to show/hide the parameters.

Parameter	Description
Bearing Type	Select from a variety of bearing types
Single Row, Separable	 Double Ribs - Inner Double Ribs - Outer, Outer Ring Separable  Double Ribs - Inner Double Ribs - Outer, Inner Ring Separable
Single Row	 Double Ribs - Inner/ No Ribs - Outer  No Ribs - Inner/Double Ribs - Outer  Double Ribs - Inner/ One Rib - Outer  One Rib - Inner/ Double Rib - Outer

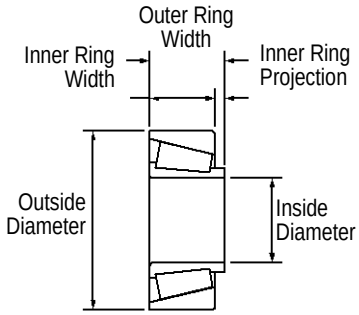
Parameter	Description
Single Row, Self Aligning	
Double Row	
Double Row, Self-Aligning	
Inside Diameter	Specify the inside diameter of the bearing
Outside Diameter	Specify the outside diameter of the bearing
Width	Enter the width of the bearing
View (2D only)	Select the 2D view 
Show Detail (2D only)	Indicate whether to display a detailed 2D view of the bearing 
Show Center Line (2D only)	Select to draw the bearing with center lines
Section (3D only)	Select to display a section of the bearing

Parameter	Description
Highlight Rollers (3D only)	Select to highlight the rollers within the bearing in black

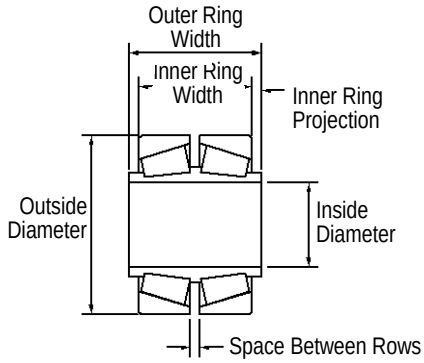
D Tapered Roller Bearings

To insert a tapered roller bearing:

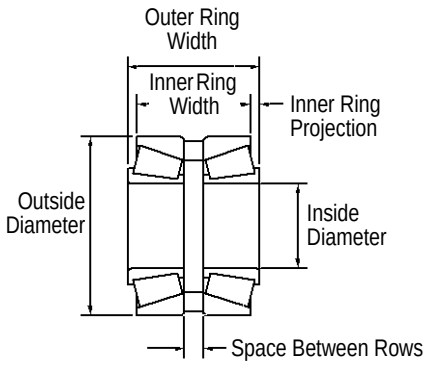
1. Click the **Tapered Roller Bearing** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Single Row (2D)



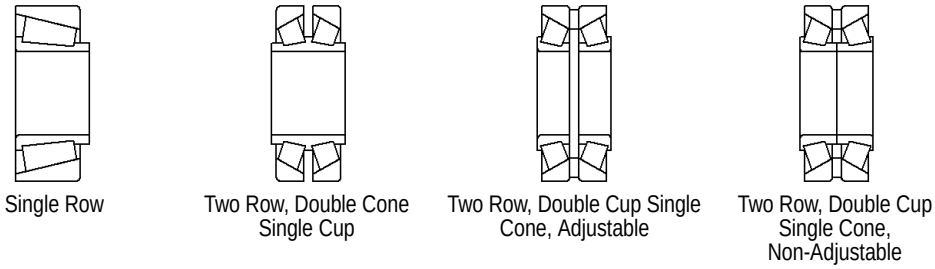
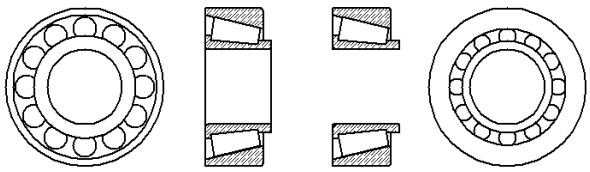
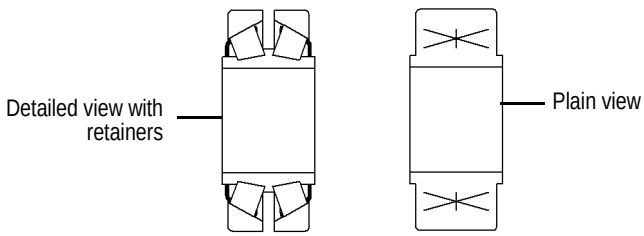
Two row, double cone, single cups (2D)



Two row, double cup, single cone (2D)

Section with rollers highlighted in black, and a retainer (3D)

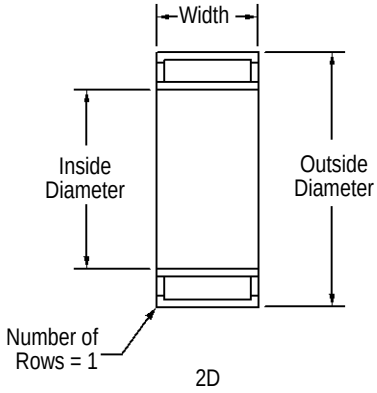
Click to show/hide the parameters.

Parameter	Description
Bearing Type	Select from a variety of bearing types  Single Row Two Row, Double Cone Single Cup Two Row, Double Cup Single Cone, Adjustable Two Row, Double Cup Single Cone, Non-Adjustable
Inside Diameter	Specify the inside diameter of the bearing
Outside Diameter	Specify the outside diameter of the bearing
Width of Inner Ring	Enter the width of the inner ring
Width of Outer Ring	Enter the width of the outer ring
Space Between Rows	Enter the space between rows
Inner Ring Projection	Specify the projection distance of the inner ring
View (2D only)	Select the 2D view  Front Section Assembly Rear
Show Retainers	Select to display the bearing with retainers; **Show Detail** must also be selected in order to view the retainers in 2D bearings
Show Detail (2D only)	Indicate whether to display a detailed 2D view of the bearing (including balls and/or retainers)  Detailed view with retainers Plain view
Show Center Line (2D only)	Select to draw the bearing with center lines
Section (3D only)	Select to display a section of the bearing
Highlight Rollers (3D only)	Select to highlight the rollers within the bearing in black

Needle Bearings

To insert a needle bearing:

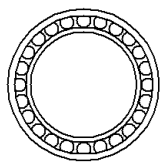
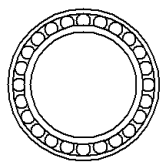
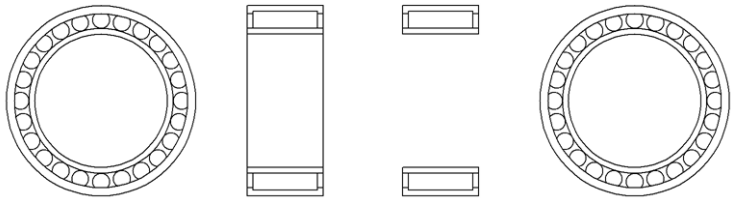
1. Click the **Needle Bearing** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



2D

3D
(Section with rollers highlighted in black)

Click to show/hide the parameters.

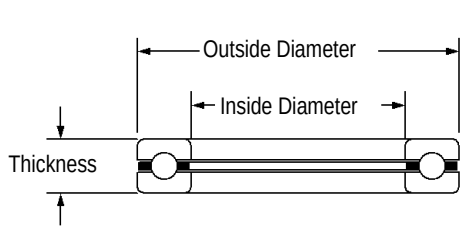
Parameter	Description
Bearing Type	Select one of the bearing types  No Ribs-Inner/Double Ribs - Outer  Double Ribs-Inner/No Ribs - Outer
Inside Diameter	Specify the inside diameter of the bearing
Outside Diameter	Specify the outside diameter of the bearing
Width	Enter the width of the outer ring
Number of Rows	Enter the number of bearing rows
View (2D Only)	Select the 2D view  Front Section Assembly Rear

Parameter	Description
Show Inner Ring	Select to display the bearing inner ring
Show Detail (2D Only)	Indicate whether to display a detailed 2D view of the bearing
Show Center Line (2D Only)	Select to draw the bearing with center lines
Section (3D Only)	Select to display a section of the bearing
Highlight Rollers (3D Only)	Select to highlight the rollers within the bearing in black

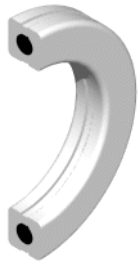
D Thrust Bearings

To insert a thrust bearing:

1. Click the **Thrust Bearing** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

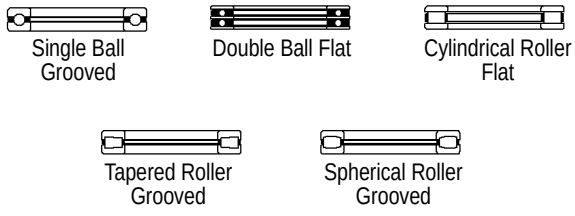


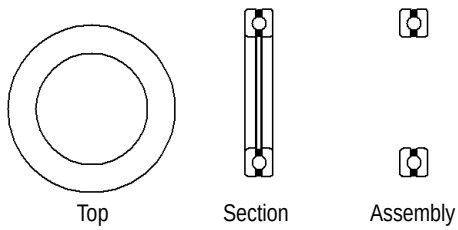
2D



3D
(Section with rollers highlighted in black)

Click to show/hide the parameters.

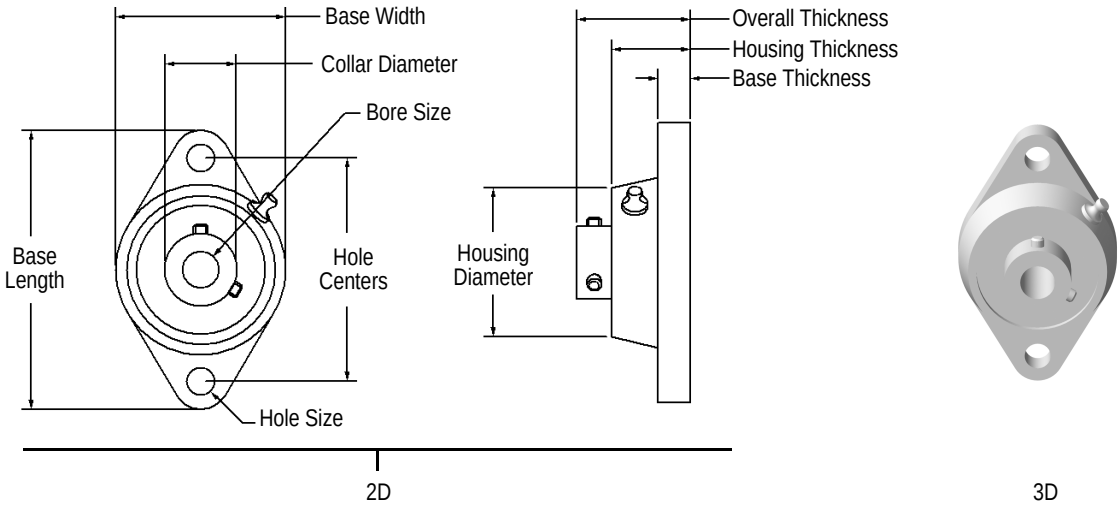
Parameter	Description	
Bearing Type	Select one of the bearing types	
Raceway Style (Single Ball, Double Ball, and Cylindrical Roller types only)	Choose either grooved or flat	
Inside Diameter	Specify the inside diameter of the bearing	
Outside Diameter	Specify the outside diameter of the bearing	
Thickness	Enter the width of the outer ring	

Parameter	Description
View (2D Only)	Select the 2D view 
Show Retainer(s)	Select to display the bearing retainer(s); **Show Detail** must also be selected to view the retainer(s) in 2D bearings
Show Detail (2D Only)	Indicate whether to display a detailed view of the 2D bearing (including balls)
Show Center Line (2D Only)	Select to draw the bearing with center lines
Section (3D Only)	Select to display a section of the bearing
Highlight Rollers (3D Only)	Select to highlight the rollers within the bearing in black

D Flanged Bearings: 2 Hole

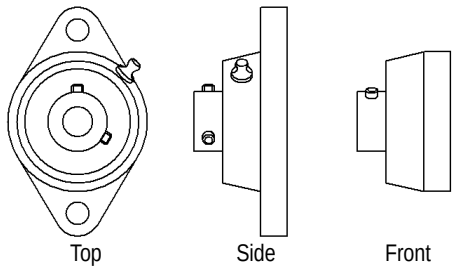
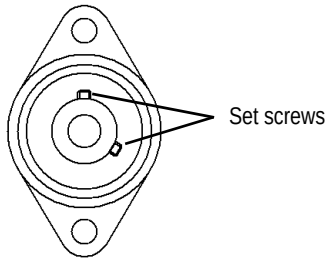
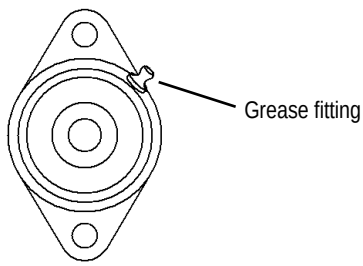
To insert a flanged bearing - 2 hole:

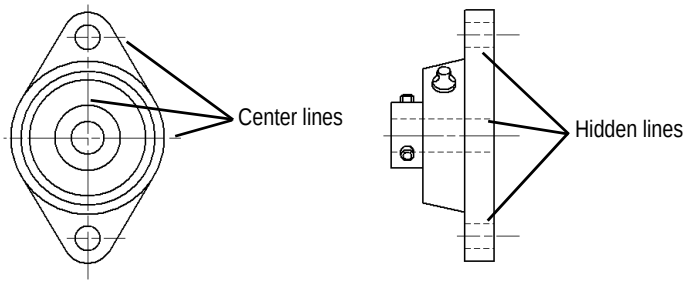
1. Click the **Flanged Bearing - 2 Hole** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Nominal Size	Select one of the common flanged bearing sizes; the associated parameters are displayed in the Object Info palette

Parameter	Description
Use Custom Dimensions	Select to apply custom parameter modifications
Bore Size	Indicate the bore diameter
Collar Diameter	Specify the diameter of the bearing collar
Housing Diameter	Indicate the diameter of the housing
Base Width	Specify the width of the base
Base Length	Specify the length of the base
Hole Centers	Indicate the distance between the hole centers
Hole Size	Indicate the diameter of the mounting holes
Base Thickness	Specify the width of the bearing base
Housing Thickness	Enter the housing thickness
Overall Thickness	Specify the width of the bearing from the top of the collar to the bottom of the base
Set Screw Size	Indicate the size of the locking set screws
View (2D Only)	Select the 2D view 
Show Set Screws	Select to display the set screws 
Show Grease Fitting	Select to display the grease fitting 

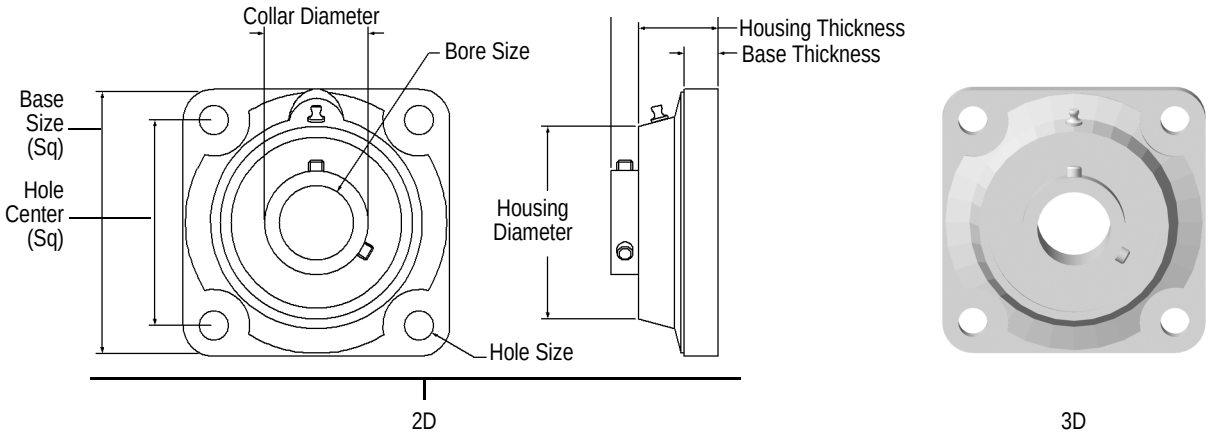
Parameter	Description
Show Center Lines and Hidden Lines (2D Only)	Select to draw the bearing with center lines and hidden lines 

Flanged Bearings: 4 Hole

Flanged Bearings: 4 Hole

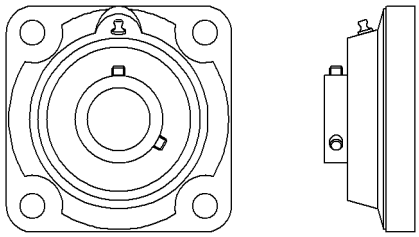
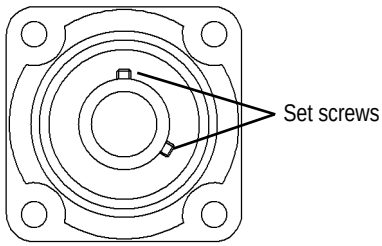
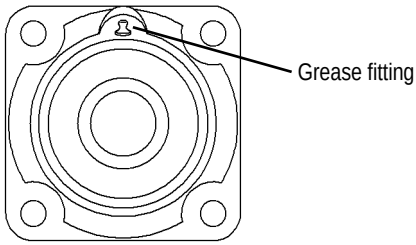
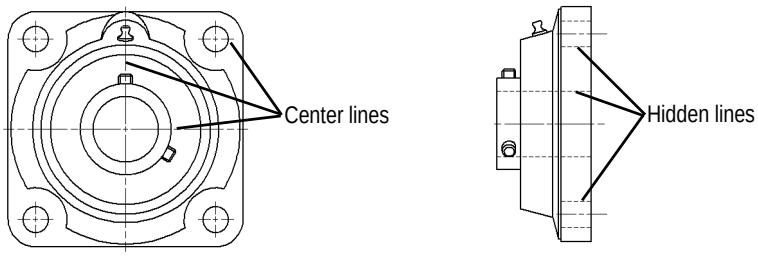
To insert a flanged bearing - 4 hole:

1. Click the **Flanged Bearing - 4 Hole** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Nominal Size	Select one of the common flanged bearing sizes; the associated parameters are displayed in the Object Info palette
Use Custom Dimensions	Select to apply custom parameter modifications
Bore Size	Indicate the bore diameter
Collar Diameter	Specify the diameter of the bearing collar
Housing Diameter	Indicate the diameter of the housing

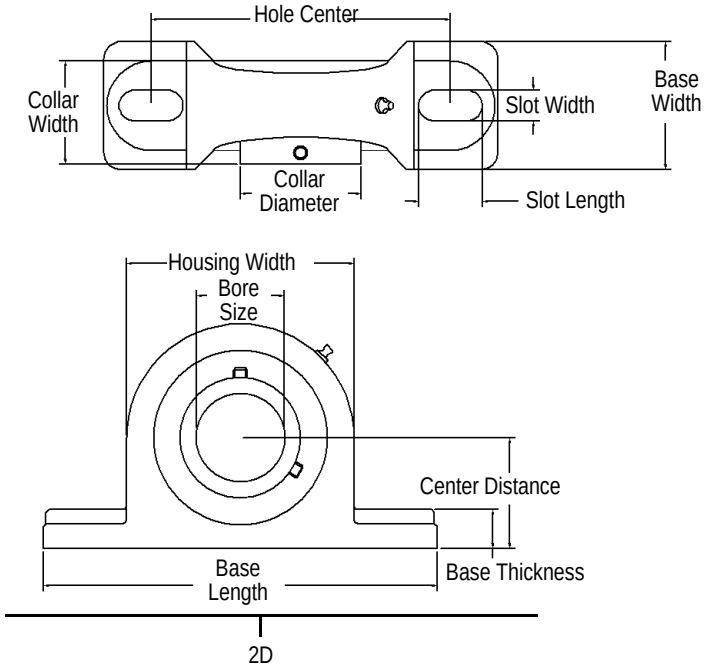
Parameter	Description
Base Size (Sq.)	Specify the dimensions of the bearing base
Hole Centers (Sq.)	Indicate the distance between the mounting hole centers
Hole Size	Indicate the diameter of the mounting holes
Base Thickness	Specify the width of the bearing base
Housing Thickness	Enter the housing thickness
Overall Thickness	Specify the width of the bearing from the top of the collar to the bottom of the base
Set Screw Size	Indicate the size of the locking set screws
View (2D Only)	Select the 2D view 
Show Set Screws	Select to display the set screws 
Show Grease Fitting	Select to display the grease fitting 
Show Center Lines and Hidden Lines (2D Only)	Select to draw the bearing with center lines and hidden lines 

Flanged Bearings: 2 Hole

Pillow Block Bearings

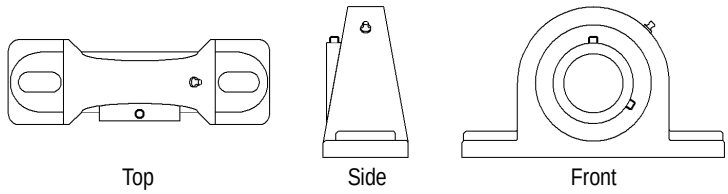
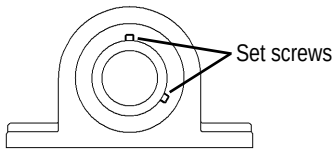
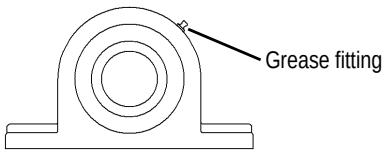
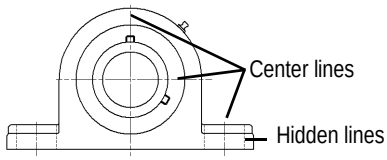
To insert a pillow block bearing:

1. Click the **Pillow Block** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

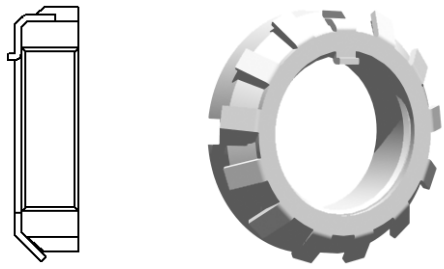
Parameter	Description
Size	Select one of the common pillow block bearing sizes; the associated parameter values are displayed in the Object Info palette
Use Custom Dimensions	Select to change any parameter values
Bore Size	Indicate the bore diameter
Center Distance	Indicate the length from the center of the bore to the bottom of the base
Housing Width	Enter the width of the housing
Base Length	Specify the length of the base
Base Thickness	Specify the thickness of the base
Hole Centers	Specify the distance between the hole centers
Slot Width	Indicate the width of the slots
Slot Length	Indicate the length of the slots
Base Width	Specify the width of the block base

Parameter	Description
Collar Width	Enter the width of the collar
Collar Diameter	Enter the diameter of the collar
Set Screw Size	Indicate the size of the locking set screws
View (2D Only)	Select the 2D view 
Show Set Screws	Select to display the set screws 
Show Grease Fitting	Select to display the grease fitting 
Show Center Lines and Hidden Lines (2D Only)	Select to draw the 2D bearing with center lines and hidden lines 

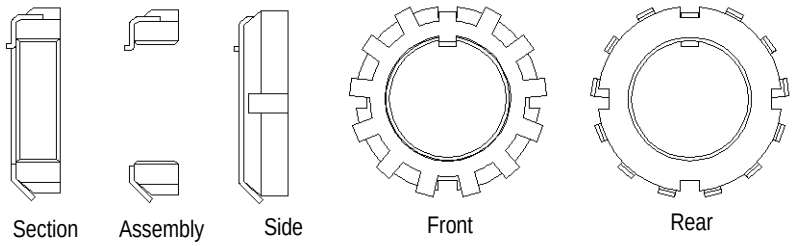
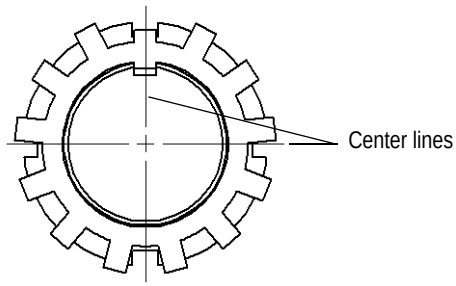
D Bearing Lock Nut

To insert a bearing lock nut:

1. Click the **Bearing Lock Nut** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

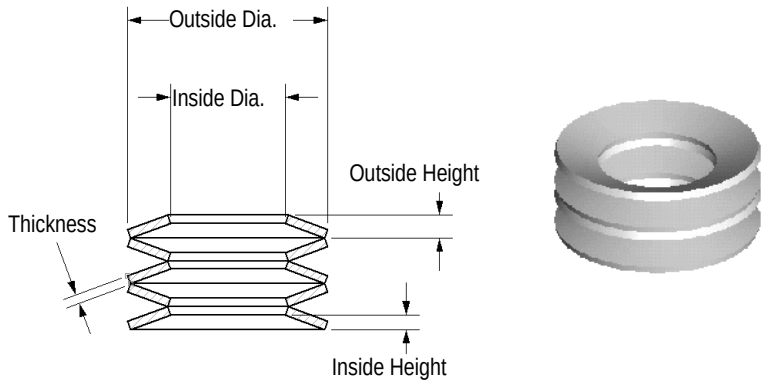
Parameter	Description
Size	Select the bearing lock nut size
View (2D Only)	Select the 2D view 
Show Washer	Select to display a washer with the bearing lock nut. By default, the washer is displayed. The side view does not display a washer.
Show Center Line (2D Only)	Select to draw the 2D bearing lock nut with center lines 

Springs

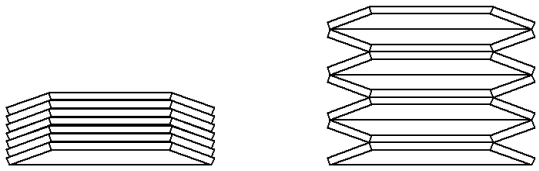
Belleville Springs

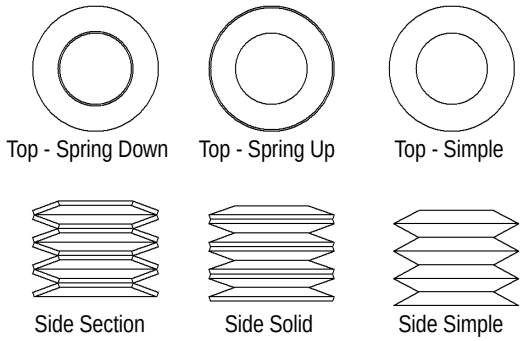
To insert a belleville spring:

1. Click the **Belleville Spring** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Inside Diameter	Specify the inside diameter of the Belleville spring
Outside Diameter	Specify the outside diameter of the Belleville spring
Thickness	Enter the wire thickness
Height	Enter the height of each spring section
Height is	Indicate whether the **Height** value applies to the **Inside** or **Outside** of the spring
Number of Springs	Specify the number of springs
Configuration	Select the spring configuration  Same Direction Alternate Down and Up
Number Down / Number Up (Alternate only)	For Alternate Down and Up spring configurations, enter the number of springs which are oriented up and the number of springs which are oriented down
Draw First Spring Up (Alternate only)	For Alternate Down and Up spring configurations, select to draw the first spring section oriented up

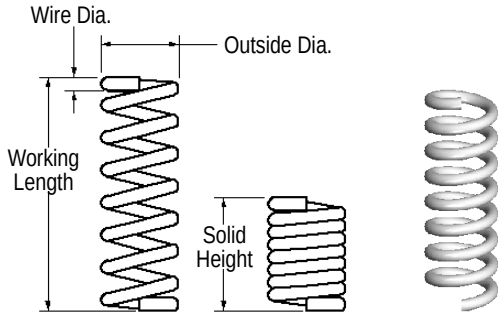
Parameter	Description
View (2D Only)	Select the 2D view 
Draw Center Line (2D Only)	Select to draw the 2D spring with center line(s)

D Compression Springs 1 and 2

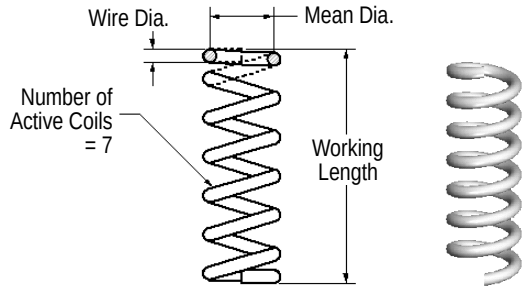
To insert a compression spring 1:

To insert a compression spring 2:

1. Click the **Compression Spring 1** or **Compression Spring 2** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.

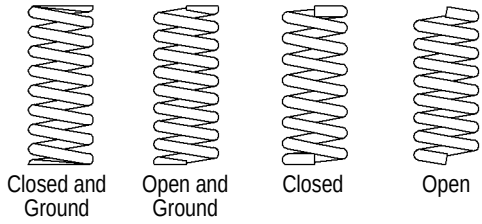
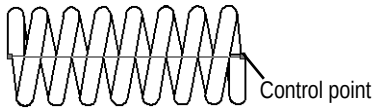
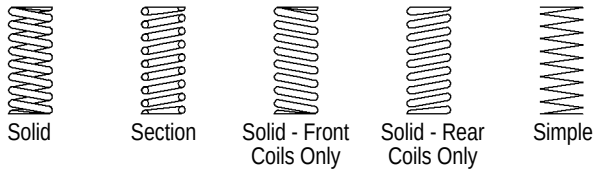


Compression Spring 1 parameters (2D) 3D



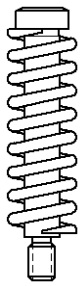
Compression Spring 2 parameters (2D) 3D

Click to show/hide the parameters.

Parameter	Description
Type of Ends	Specify the spring end style 
Outside Diameter	For Compression Spring 1, enter the outside diameter
Mean Diameter	For Compression Spring 2, enter the mean diameter
Wire Diameter	Enter the wire diameter
Solid Height	For Compression Spring 1, enter the solid height (compressed height)
Number of Active Coils	For Compression Spring 2, specify the number of active coils
Working Length	Either specify the spring working length value, or click and drag a spring control point to define the length graphically 
Spiral	Select either a **Left Hand** or **Right Hand** spiral direction
Display (2D Only)	Select a display mode for 2D springs 
Draw Center Line (2D Only)	Select to draw the spring with center line(s)

A utility calculates spring rates for compression springs (see “Spring Calculator” on page 1824).

To simulate the appearance of a spring wound around an object (such as a shaft or screw), use two springs. Set the front spring’s display parameter to **Solid - Front Coils Only**. The spring behind the object is set to **Solid - Rear Coils Only**. Adjust the objects’ relative position with the **Modify > Send > Send to Front** and **Modify > Send > Send to Rear** commands until the object displays correctly.



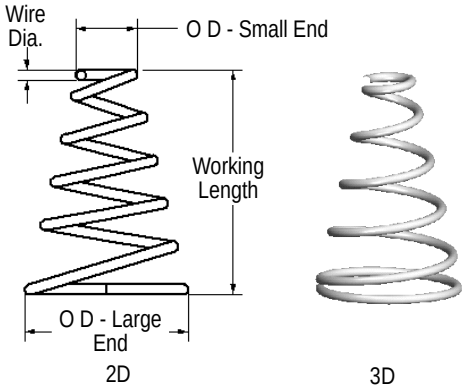
Conical Compression Springs

D Conical Compression Springs

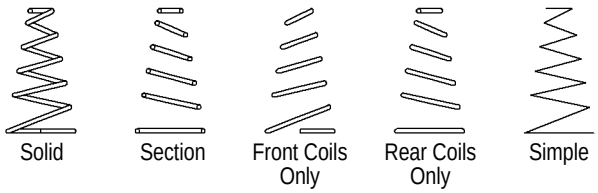
To insert a conical compression spring:

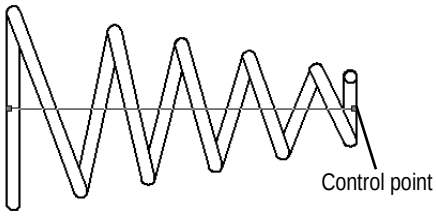
1. Click the **Conical Compression Spring** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. Conical compression springs have ends of unequal diameter. When drawing a conical compression spring, the first click defines the large end of the spring, and the second click defines the small end.

   If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Display (2D Only)	Select a display mode for 2D springs 
Wire Diameter	Enter the wire diameter (thickness)
OD - Small End	Specify the outer diameter of the small end of the spring
OD - Large End	Specify the outer diameter of the large end of the spring
No of Active Coils	Specify the number of active coils in the spring

Parameter	Description
Working Length	Either specify the spring working length value, or click and drag a spring control point to define the length graphically 
Spiral	Select either a **Left Hand** or **Right Hand** spiral direction
Draw Center Line (2D Only)	Select to draw the spring with center line(s)

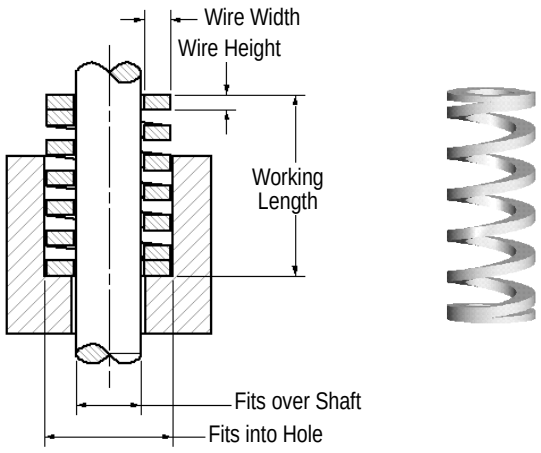
Compression Springs 1 and 2

Die Springs

Die springs are made of rectangular wire.

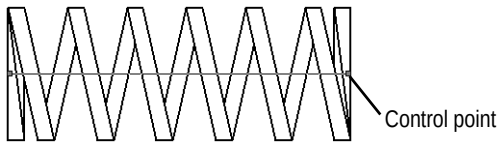
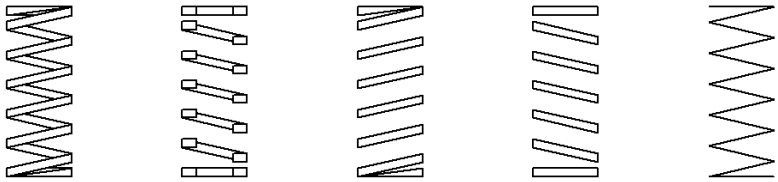
To insert a die spring:

1. Click the **Die Spring** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



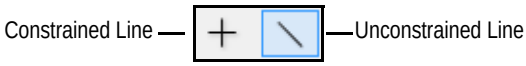
Click to show/hide the parameters.

Parameter	Description
Fits into Hole	The spring will be able to fit into a hole of this diameter

Parameter	Description
Fits over Shaft	The spring will be able to fit over a shaft of this diameter
Wire Width	Specify the width of the wire
Wire Height	Enter the height of the wire
Solid Height	Enter the height of the spring at maximum compression
Working Length	Either specify the spring working length value, or click and drag a spring control point to define the length graphically 
Spiral	Select either a **Left Hand** or **Right Hand** spiral direction
Display (2D Only)	Select a display mode for 2D springs 
Draw Center Line (2D Only)	Select to draw the spring with center line(s)

Extension Springs: Front View and 3D

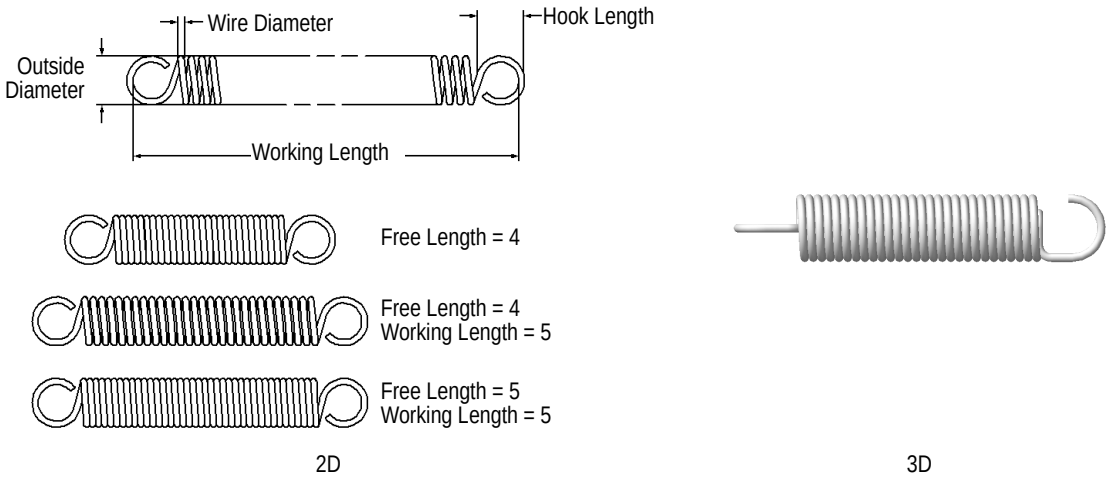
In front view and 3D, two modes are available.



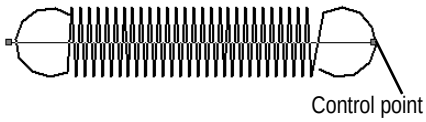
Mode	Description
Constrained Line	Lines are drawn at 30°, 45°, and 90° angles, and their complements are drawn in increments of 30° and 45°
Unconstrained Line	Lines can be drawn at any angle; press the Shift key to temporarily constrain a line during drawing

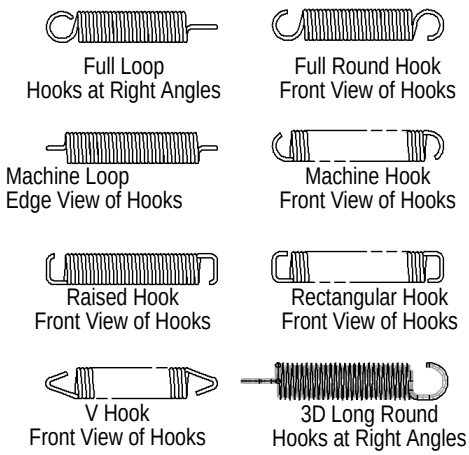
To insert an extension spring - front view:

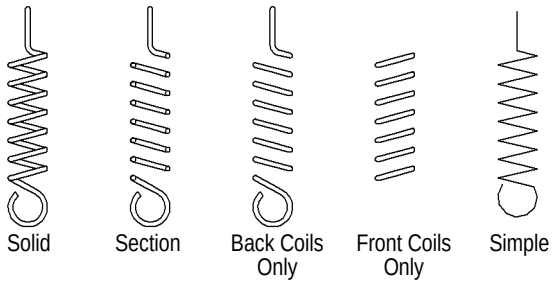
1. Click the **Extension Spring - Front View** tool or the **Extension Spring - 3D** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

Parameter	Description
Outside Diameter	Specify the outside diameter of the coiled section of the spring
Wire Diameter	Enter the wire diameter
Free Length	Length of spring when not extended (no force is applied)
Working Length	Either specify the spring working length value, or click and drag a spring control point to define the length graphically 
End Style	Select a style for the ends of the spring
Hook Length	Enter the length of the hook from the spring end to the top of the hook
Hooks at Right Angles (3D only)	Select the create the 3D hooks at a right angle to each other
Spiral	Select either a **Left Hand** or **Right Hand** spiral direction
View (Front Only)	Select the 2D hook's view

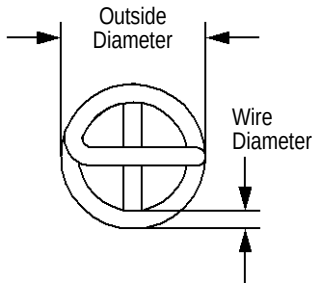


Parameter	Description
Display (Front Only)	Select a display mode for the 2D spring 
Set Free Length to Working Length (Front only)	Select to force the spring's **Free Length** value to equal the **Working Length** value; this shows the coils tightly wound at any working length
Show All Coils (Front only)	Select to draw all of the spring's coils
Draw Center Line (Front only)	Select to draw the spring with center line(s)

D Extension Springs: End View

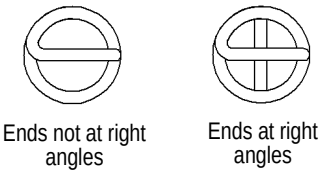
To insert an extension spring - end view:

1. Click the **Extension Spring - End View** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



Click to show/hide the parameters.

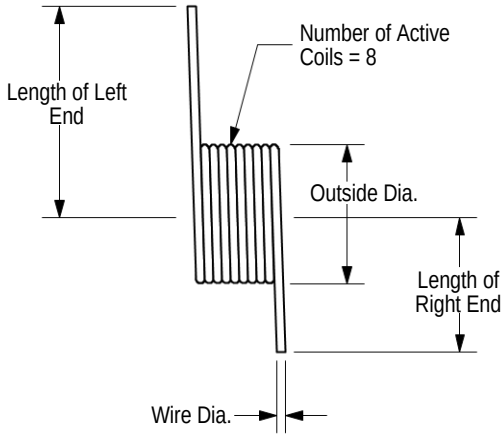
Parameter	Description
Wire Diameter	Enter the wire diameter
Outside Diameter	Specify the outside diameter of the coiled section of the spring

Parameter	Description
Ends at Right Angles	Indicate whether the spring ends are displayed at right angles to each other 
Spiral	Select either a **Left Hand** or **Right Hand** spiral direction
Draw Center Lines	Select to draw the spring with center lines

Torsion Springs: Front View

To insert a torsion spring - front view:

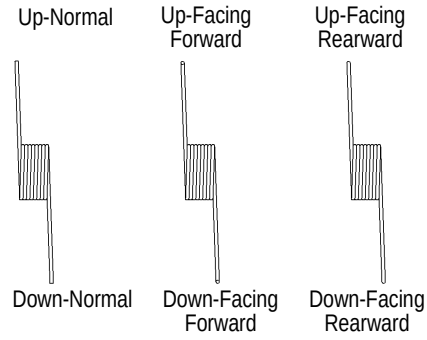
1. Click the **Torsion Spring - Front View** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



In the Object Info palette, set the torsion spring properties (**Outside Diameter**, **Wire Diameter**, **Number of Active Coils**, and **Spiral**), and then set the properties for each end of the spring.

Click to show/hide the parameters.

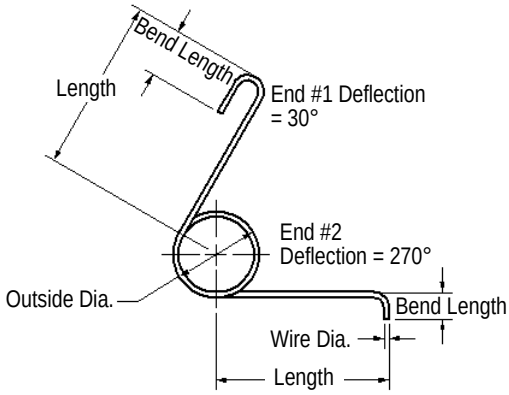
Parameter	Description
Outside Diameter	Specify the outside diameter of the coiled section of the torsion spring
Wire Diameter	Enter the wire diameter
Number of Active Coils	Specify the number of active coils in the spring
Left/Right End	Select to enter the dimensions for left and/or right end
Length	For each end, specify the length between the coil center and the end

Parameter	Description	
Direction	Indicate whether each end is facing up or down	
Orientation	Select forward, rearward, or normal for each end	
Spiral	Select either a **Left Hand** or **Right Hand** spiral direction	

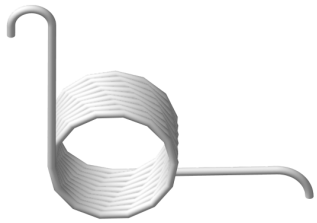
Torsion Springs: End View and 3D

To insert a torsion spring - end view:

1. Click the **Torsion Spring - End View** tool or the **Torsion Spring - 3D** tool from the Machine Components tool set.
2. Click to place the object in the drawing, and click again to set the object's rotation. If this is the first time the object is placed in the drawing, an object properties dialog box opens. These parameters apply to subsequently created objects; they can be changed later by accessing them from the Object Info palette.
3. Specify the object properties and click **OK**.



2D

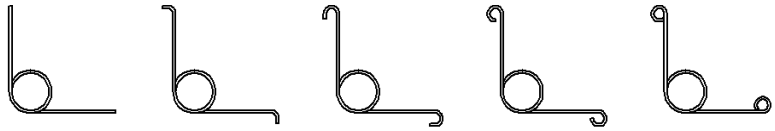


3D

In the Object Info palette, set the torsion spring properties (**Wire Diameter**, **Outside Diameter**, and **Spiral**), and then set the properties for each end of the spring.

Click to show/hide the parameters.

Parameter	Description
Wire Diameter	Enter the wire diameter
Outside Diameter	Specify the outside diameter of the coiled section of the torsion spring

Parameter	Description
Number of Active Coils (3D only)	Specify the number of active coils in the coiled section of the 3D torsion spring
End # 1/2	Select to enter the dimensions for End #1 and/or End #2
Length	For each end, specify the length between the coil center and the bend
Deflection	Enter the deflection angle for each end
End Style	Select a style for each end (specify a clockwise (CW) or counterclockwise (CCW) direction)  Plain 1/4 Bend 1/2 Bend 3/4 Bend Full Loop
Bend Radius	For each end, specify the radius value of the bend (does not apply to **Plain** End Styles)
Bend Length	For each end, specify the length between the bend and the wire end (does not apply to **Plain** or **Full Loop** End Styles)
Spiral	Select either a **Left Hand** or **Right Hand** spiral direction

Solar Studies

The **Heliodon** tool, available in the Vectorworks Design Series, places one or more special directional lights in a drawing for conducting sun studies, calculating shadow angles, and creating solar animations. The tool demonstrates the location of sunlight and shadows cast on the model as the sun's position changes, for any day of the year and any location on earth. Accurate region/city information is used for the calculations. Vertical and horizontal shadow angles can be automatically calculated for determining the correct placement of shading devices, such as awnings. When combined with a physical sky background, the heliodon can create natural lighting that responds to the sun position and time of day. Additionally, an area of a drawing, such as a window or a portion of a large model, can be studied to determine when (or if) sunshine will affect it. Solar animations can be generated on screen or by exporting to a movie file.

To show shadows, the Renderworks product is required; to show soft shadows, a Renderworks render mode is required.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting a Heliodon Object

To insert a heliodon:

1. Click the **Heliodon** tool from the Visualization tool set.

   If this is the first time a heliodon has been placed on the drawing, the Settings dialog box opens when placing the object. Specify the preferences to use for this tool during this session. (Otherwise, click **Preferences** from the Tool bar to specify or change the tool parameters.)

Click to show/hide the parameters.

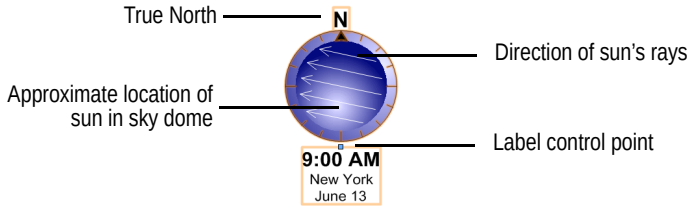
Parameter	Description
Time Display	Select the method of time display; **Samples** shows the time as it is currently formatted
12/24 Hour	Select whether to display the time in 12-hour or 24-hour time format; in 12-hour format, the date automatically displays as Month/Day, and in 24-hour format, the date automatically displays as Day/Month
Colon/Dot	Sets the time to display with a colon or dot between the hour and minute
2D Graphic	Select a heliodon symbol from the default content; default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see "Resource Libraries" on page 219
Location	Locations are derived from text files that can be found in the heliodon cities file in the [Vectorworks]\Libraries folder (see "Resource Libraries" on page 219). Region text files with city, longitude, and latitude information can be added to the current set of files, or the information can be edited from this dialog box. Once a region has been selected, the corresponding text file is saved to the user folder; see "User Folders Preferences" on page 57.
Region	Select the region from among the major regions of the world, or click Add New Region from the bottom of the **Region** list, to create a new region
City	Select a major city from within the specified region
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Edit City	If you have different information for the currently selected city, its information can be edited; new cities can also be added to the current region. Any edits are saved to the user folder.
City	If updating the currently selected city, do not edit the city name; if adding a city, enter the new city name
Latitude/Longitude	Specifies the city's global coordinates
North/South/East/ West	Sets the latitude (North/South) and longitude (East/West) hemispheres
Time Zone	Select the city's time zone based on Greenwich Mean Time (GMT)
Add	Adds the new city to the selected region; the new city must have a unique name
Update	Updates the information of the selected city
Delete	Deletes the city from the selected region

2. Click **OK**.
3. Click once in the drawing to set the object's position. Click again to set the object's rotation.

   The heliodon graphic displays true North at the top of the heliodon. Normally, the drawing is oriented to page North (true North, not magnetic North), and true North matches the top of the page, with no compensation required for proper sun position. If the drawing was not created with this orientation, use the Object Info palette to specify a **Rotation** in degrees from page North. North set here should match true North in the drawing.

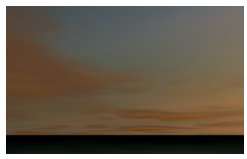
   The white arrows represent the sunlight direction at the current time of day and the designated location. The gradient highlight shows the approximate position of the sun in the sky dome.



4. Edit the properties of the heliodon in the Object Info palette, as needed.

Click to show/hide the parameters.

Parameter	Description
Settings	Opens the Settings dialog box, to specify the heliodon parameters
Heliodon data	Displays the location, position, and time zone set for the heliodon
Show Solar Data	Displays solar information for the heliodon in the Object Info palette; select a **Text Display** option which includes "Data" to include the solar data in the heliodon text label
Daylight Savings	Select if Daylight Saving Time is in effect
Use Solar Time	Overrides the local time zone and daylight savings settings, using apparent solar time instead of mean solar time. When solar time is used, Daylight Saving Time cannot also be in effect.

Parameter	Description
Time/Day/Month	Specify the time of day, the day of the month, and the month, or select a winter or summer solstice or spring or autumn equinox from the **Month** list. The **Time** can also be specified as sunrise, dawn, midday, noon, dusk, sunset, or midnight.
Sun Brightness	Sets the brightness of the directional light representing the sun
Renderworks Only	Renderworks is required for the four next parameters
Soft Shadows	Creates soft shadows when rendering shadows with a Renderworks mode
Physical Sun	Creates a physical sun with color and brightness that is controlled by the heliodon settings (when rendering with a Renderworks mode); deselect the option to use a white directional light, set to 100% brightness
Physical Sky	Allows the heliodon settings, such as date, time, and use of warm colors, to control a physical sky Renderworks background, when one is applied (see “Creating Physical Sky Backgrounds” on page 1526). This creates more realistic solar animations, because the sky reflects the time of day and month of the year as the animation proceeds.  July 13, 6:00 am  July 13, 3:00 pm  July 13, 8:30 pm Deselect this option to view the physical sky with bright, midday settings.
Warm Colors	Adds a rosier, richer appearance to physical sun and sky colors  Warm Colors selected  Warm Colors deselected
Show Shadow Angle Calculator	Displays additional options for calculating shadow angles for selected walls
Object Display	Select how to display the heliodon object in 2D and 3D views; the heliodon symbol is a 2D graphic. • **2D Graphic + 3D Light:** Displays as a 2D graphic in Top/Plan view and as a light in 3D views • **2D Graphic Only:** Displays as a planar graphic in all views • **3D Light Only:** Displays as a light object; does not use a 2D graphic.

Parameter	Description
Text Display	Select the information to include in the heliodon's text label; move the label control point to adjust the label position. • **None:** Does not display a text label • **Time + Location + Date:** Displays the time, city, and date • **Time + Location + Date + Data:** Displays the time, city, date, and solar data including solar time • **Time + Date Only:** Displays the time and date • **Time + Date + Data**: Displays the time, date, and solar data • **Data Only:** Displays the solar data
Solar Animation	Opens the Solar Animation dialog box; see "Creating Solar Animations" on page 1492

Once placed, a heliodon symbol is added to the drawing file and is listed in the Resource Browser in the Heliodon Symbols folder. Edit the 2D component of the symbol as described in "Editing Symbol Definitions" on page 247 to change the appearance or the font style or size.

Heliodon objects display in, and can be controlled from, the Visualization palette. They can be identified by their location and time data.

> Place several heliodon objects in the drawing, set to different times of day, to conduct a solar study. Turn the heliodon objects on and off in the Visualization palette to study the effects of the sun's light and shadows cast at different times of day.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Solar Studies
Creating Solar Animations
Managing Lights and Cameras with the Visualization Palette
Calculating Shadow Angles
Creating Physical Sky Backgrounds

D Creating Solar Animations

A solar animation illustrates the way the sun moves over, or shadows are cast upon, a project as the sun's position changes. It studies how the sun affects a model or site at a particular time of day or year. The animation can be set to rotate the sun about the model, or the model can rotate about the layer plane center or heliodon object.

Solar animations can be interactive, observing the effects in the drawing as changes are made. To interactively view changes as you move the sliders, render in Wireframe or OpenGL render mode. The solar animation can also be exported to a .mov file.

Depending on the view, render mode, presence of shadows, and heliodon view, various solar assessments can be conducted. Generally, OpenGL rendering (with or without shadows; Renderworks required) will provide good results.

Settings	Solar Animation Effect
Wireframe, Top/Plan view	Observe the heliodon symbol while changing the time of year or time of day. Animating shows the position of the sun's rays and approximate position of the sun in the sky dome. The date display on the heliodon label changes to indicate the animation's day of the month.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Settings	Solar Animation Effect
OpenGL rendering (**Use Shadows** not selected), Top/Plan or 3D view	Observe the model as the sun's rays move over it during the course of the year or day. Different portions of the model are illuminated by sunlight.
OpenGL rendering with shadows (**Use Shadows** selected), Top/Plan view or 3D view	Observe the shadows cast by the model over the course of the day or year, and observe the portions of the model that are illuminated by sunlight.
Solar animation view set to **View (sun to layer plane center)** or **View (sun to Heliodon center)**	Observe as the model rotates, as if the viewer was positioned on the sun. Any visible surface has sunlight falling on it, while anything that can't be seen is in shadow (therefore, **Use Shadows** is not necessary). The model rotates about the layer plane center or about the heliodon center. Rotating about the heliodon center can be useful when the model is not centrally located, or when a particular part of the model needs to be studied. For example, place the heliodon adjacent to a shading device, zoom in, and then animate using the heliodon center.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Interactive Solar Animations
Exporting Solar Animations
Solar Studies
Inserting a Heliodon Object
Calculating Shadow Angles

Creating Interactive Solar Animations

Interactive solar animations display the sun's effects within the drawing; specify the view, day, month, and time for interactive previews. Wireframe mode views as well as rendered views (with or without shadows) are useful to study the sun's movements.

Large or complex rendered models, high-quality render settings, and smooth shadows may affect the animation. To interactively view changes as you move the sliders, render in Wireframe or OpenGL render mode. Low to medium (**Detail** and **Quality**) OpenGL options are recommended.

To animate the sun interactively:

1. Insert at least one heliodon object as described in "Inserting a Heliodon Object" on page 1489.
2. Select the desired view and render mode.
3. Select the heliodon, and click **Solar Animation** from the Object Info palette. Alternatively, double-click on a heliodon object.

   The Solar Animation dialog box opens. Click on the Interactive tab. Set the **Animate** view, and the day, month, and time, or move the month and day sliders while observing the effects.

Click to show/hide the parameters.

Parameter	Description
Animate	Sets whether to animate the sun about the model, or rotate the model about the sun
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Heliodon (Top/Plan view) or Sun (using current view)	Moves the sun over the model
View (sun to layer plane center)	Navigates around the model as viewed from the sun, showing the part of the model illuminated by the sun at that time; the model is assumed to be at the layer plane center and rotates about this center
View (sun to Heliodon center)	Navigates around the heliodon as viewed from the sun; parts of the model that are visible are the areas receiving sunlight. This mode is helpful when studying a portion of a larger model or when the model is offset from the layer plane center. Place the heliodon in the area of interest.
Interactive/Export Movie	Select whether to view the animation interactively, within the drawing, or export it to an animation file
Day/Month	Enter the day of the month and the month, either by typing digits or using the up and down arrows
Month slider	Move the interactive slider to progress through the months of the year
Hour/Min	Enter the time as hour and minute, either by typing digits or using the up and down arrows. The increment for the minute arrows is set by the **Minute Interval**.
AM/PM (12 Hour time display only)	Toggles between AM and PM for a quick shortcut to setting morning or afternoon
Day slider	Move the interactive slider to progress through the daylight hours
Minute Interval	Sets the increment to advance or decrease the minutes with the up/down arrows, making it easier to quickly step through the day if needed.
Local Time/Solar Time	Displays the sunrise and sunset times for the location and time of year, as well as the current local or solar time. The display of local or solar time depends on whether **Use Solar Time** is selected in the Object Info palette.

4. When finished with the solar animation, click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Exporting Solar Animations
Creating Solar Animations
Inserting a Heliodon Object
Solar Studies

Exporting Solar Animations

The same type of solar animation effects that can be seen in the drawing in interactive mode can be exported to a movie.

To export a solar animation:

1. Insert at least one heliodon object as described in “Inserting a Heliodon Object” on page 1489.
2. Select the desired view and render mode.
3. Select the heliodon, and click **Solar Animation** from the Object Info palette. Alternatively, double-click on a heliodon object.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Solar Animation dialog box opens.

4. Click on the Interactive tab, and set the date and time.
5. Click on the Export Movie tab. Set the **Animate** view and the export options. The solar animation occurs based either on time or date settings.

Click to show/hide the parameters.

Parameter	Description
Animate	Sets whether to animate the sun about the model, or rotate the model about the sun
Heliodon (Top/Plan view) or Sun (using current view)	Moves the sun over the model
View (sun to layer plane center)	Navigates around the model as viewed from the sun, showing the part of the model illuminated by the sun at that time; the model is assumed to be at the layer plane center and rotates about this center
View (sun to Heliodon center)	Navigates around the heliodon as viewed from the sun; parts of the model that are visible are the areas receiving sunlight. This mode is helpful when studying a portion of a larger model or when the model is offset from the layer plane center. Place the heliodon in the area of interest.
Interactive/Export Movie	Select whether to view the animation interactively, or export it to an animation file
Animate Time	Creates the solar animation over the course of a day; set the start/end time and time interval
Animate Date	Creates the solar animation over the course of a year; set the date start and end and date interval
Start Time/Date	Select the starting time or month
End Time/Date	Select the ending time or month
Time/Date Interval	Select the increment for the movie; a smaller increment creates a smoother movie, but creates a longer movie
Show time of day	Displays the time of day in the movie, at a location specified before export; set the font size
Show month and day	Displays the day and month in the movie, at a location specified before export; set the font size
Export summary	Displays the length of the movie at 15 frames per second (fps) and the number of frames to be created

6. Click **OK** to create the animation.

   If showing the time of day or month and day, click in the drawing, when prompted, to set the text location.

   The Set Movie Name and Destination dialog box opens.
7. Specify the location to save the movie file. By default, the file is named with the location and the date. Click **Save**.

   The Compression Settings dialog box opens.
8. Set the **Frames per second** and **Quality** for the movie file.
9. Click **OK**. A message displays the progress of the export, and the movie is created.

   Click to end the export early. The solar animation movie is created up to that point.

10. View the movie in the application of your choice to watch the solar animation.

Creating Interactive Solar Animations
Creating Solar Animations
Inserting a Heliodon Object
Solar Studies

D Calculating Shadow Angles

Shadow angles can be calculated and displayed for up to four walls of a model. These calculations are useful for designing or positioning shading devices, such as awnings and overhangs.

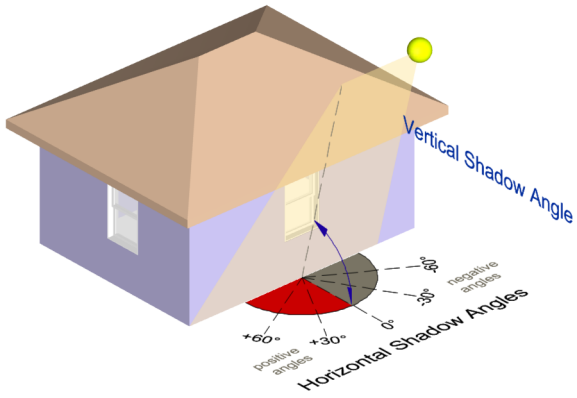
Shadow angles are always calculated relative to a wall and not to true North.

To calculate shadow angles:

1. Name the wall or walls that will participate in the calculations by selecting each wall, and then clicking on the Data tab of the Object Info palette. Enter a **Name** for the wall to identify it.
2. Insert at least one heliodon object as described in “Inserting a Heliodon Object” on page 1489.
3. Select the heliodon. In the Object Info palette, select **Show Shadow Angle Calculator**.
4. Specify the **No. of Walls** to display shadow angle calculation values. Up to four walls can be specified.
5. In **Wall Face**, specify which side of the walls, left or right, is the external side.

   By default, the wall direction as it was drawn determines which side of the wall is considered the outside and which is the inside. See “Wall Direction” on page 506.
6. Enter the exact **Name** of each wall.

   When the wall name is recognized, its Horizontal Shadow Angle (HSA) and Vertical Shadow Angle (VSA) calculated values are displayed in the Object Info palette. If a wall is always in shadow, the text **No sun on wall** displays in the Object Info palette.



Solar Studies

Sketch Rendering

The Vectorworks Design Series products include a sketch rendering mode that applies a hand-drawn or sketch effect to 2D and 3D objects in any projection. The Renderworks product is not required. The vector-based sketch effects are saved as editable resources, and are applied directly to the objects in a drawing. The sketch effects can also be applied to hatches and sheet layer viewports, and can be used in hidden line rendering.

Unlike Artistic Renderworks, sketch rendering can be applied to 2D objects, and individual objects can have specific sketch styles. Sketch effects cannot be applied to worksheets, text, loci, or lights.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

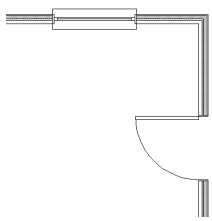
Setting the Default Sketch Style

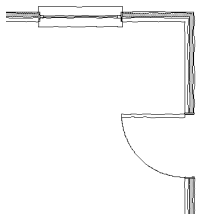
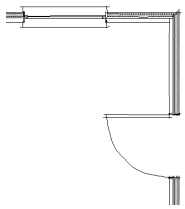
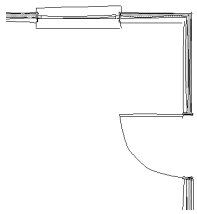
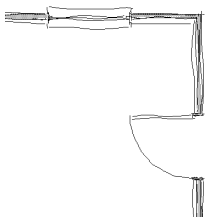
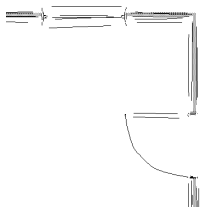
To quickly apply a sketch style to all objects in the current design or sheet layer, specify a default sketch style. Specifying the sketch style in a design layer applies the default to all layers in the file (design and sheet). For sheet layers, applying a default sketch style applies only to objects placed on the sheet layer, overriding the document default sketch style; viewports also have individual sketch style settings (see “Applying Sketch Styles to Viewports” on page 1502).

Sketch styles are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219). The following sketch styles are pre-defined, and can be selected as a default sketch style. Custom sketch styles that have been added to the file also display in the Resource Browser and can be selected as a sketch style (see “Creating Custom Sketch Styles” on page 1500).

Unused sketch styles can be purged; see “Purging Items from a File” on page 1011.

Sketch resources do not display in the Resource Browser until a sketch-related option is selected for the first time.

Default Sketch Style	Description
No Sketch	Does not apply a sketch style; useful for overriding an applied sketch style (see “Applying Sketch Styles to an Object” on page 1501) 
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Default Sketch Style	Description
Careful	Uses many small wobble points and minor variations from the actual endpoints 
Certain	Uses many small wobble points and specifies more overstrike 
Quick	Uses few wobble points, with more distance between them 
Rough	Uses many wobble points 
Tentative	Uses many small wobble points and a high amount of understrike 

To specify the default sketch style:

1. Select **View > Rendering > Sketch Options**.

   The Sketch Options dialog box opens.
2. Select the default sketch style from the list.

Click to show/hide the parameters.

Parameter	Description
Document Default Sketch Style or Sheet Layer Default Sketch Style	Specifies the sketch style to apply when the **Sketch** command is selected
Edit	Opens the Sketch Style Editor dialog box, for changing the selected sketch style parameters
Preview	Temporarily applies the current sketch style to objects in the current layer

3. Click **OK** to set the default sketch style.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying the Default Sketch Style
Editing Sketch Styles

Applying the Default Sketch Style

To apply the default sketch style to the current design or sheet layer:

Select **View > Rendering > Sketch**.

Rough sketch style depicted

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Default Sketch Style
Editing Sketch Styles
Creating Custom Sketch Styles
Applying Sketch Styles to an Object

Editing Sketch Styles

Sketch styles can be edited and renamed. The changes are saved as resources in the file.

Sketch resources do not display in the Resource Browser until a sketch-related option is selected the first time.

To edit a sketch style:

1. Select **View > Rendering > Sketch Options**.

   The Sketch Options dialog box opens.
2. Select the sketch style to edit from the sketch style list.
3. Click **Edit**.
4. The Sketch Style Editor dialog box opens. You can edit the sketch edge and end point settings, and rename the style. The preview graphic provides help with editing.

Click to show/hide the parameters.

Parameter	Description
Name	Changes the default sketch style name, if desired
Edge Settings	
Wobble	Controls how much the sketched lines are offset from the true lines of an object; drag the slider to the right to increase the wobble
Stroke	Controls the length of the sketch drawing strokes; a longer stroke distance increases the number of wobble points. Drag the slider to the right to increase the number of wobble points.
Randomness	Controls the amount of wobble and stroke variation
End Point Settings	
Overstrike	Extends sketched lines past their endpoints; drag the slider to the right to increase the overstrike
Understrike	Shortens sketched lines before their endpoints; drag the slider to the right to increase the understrike
Randomness	Controls the amount of overstrike and understrike variation

5. Click **Save**.

   Alternatively, sketch styles can be edited by selecting the style in the Resource Browser and selecting **Edit** from the context menu. Sketch styles can also be deleted with the Resource Browser. For more information on the Resource Browser, see “Using the Resource Browser” on page 221.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying the Default Sketch Style
Creating Custom Sketch Styles

Creating Custom Sketch Styles

The Vectorworks program includes several default sketch styles, but custom sketch styles can also be created

To create a custom sketch style:

1. In the Resource Browser, select **Resources > New Resource** to display the list of new resource types.

   For more information on the Resource Browser, see “Using the Resource Browser” on page 221.

2. Select **Sketch Style**.

   The Sketch Style Editor dialog box opens. Specify the sketch style settings as described in “Editing Sketch Styles” on page 1499.

3. Click **Save** to create the sketch style. The new style is listed in the Resource Browser and is available in sketch style lists.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Sketch Styles
Setting the Default Sketch Style
Applying the Default Sketch Style

Applying Sketch Styles to an Object

Each object can have its own sketch style, overriding the sketch style applied to the current layer. Apply the "No Sketch" style to objects that should never be sketched.

To apply a sketch style to an object:

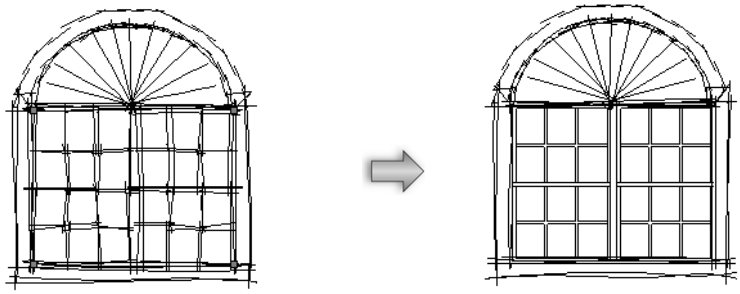
1. Select the object or objects.

   Groups and symbols cannot have a sketch style applied directly; edit the group or symbol first.

2. On the Render tab of the Object Info palette, the **Sketch** list displays the current document default sketch style. Select the desired sketch style for the object(s).

   Alternatively, select the desired sketch style resource from the Resource Browser and select **Apply** from the resource context menu, or drag the sketch style from the Resource Browser to the object.

3. The selected object's sketch style overrides the default sketch style.



Window muntins are selected

The No Sketch style is applied only to the muntins

Objects that override the default design layer sketch style also override the default sketch style on a sheet layer.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Default Sketch Style
Applying the Default Sketch Style

Applying Sketch Styles to Hatches

There are several ways to create a unique look for drawing objects. In addition to the current layer's default sketch style and the object sketch style, each hatch can have a unique sketch style.

To specify a hatch sketch style:

1. Select **Modify > Hatch**.

   The Hatches dialog box opens. For more information on hatches, see "Defining Hatches" on page 1105.

2. Select the desired hatch to edit or click **New** to create a new hatch.

   The Edit hatch dialog box opens.

3. The document default sketch style is displayed in the **Sketch Style** list. Select the desired sketch style instead, or select No Sketch to never apply a sketch style to the hatch.

   When the document default sketch style is applied, the hatch sketch style changes when the document default sketch style changes.

4. Click **OK** to exit the hatch editor.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting the Default Sketch Style

Applying Sketch Styles to Viewports

Sheet layers can have a document default sketch style applied, which changes the appearance of objects on the sheet layer, but does not sketch the viewports on the sheet layer. Each viewport can have its own sketch style, making it easy to present designs with different sketched looks.

To apply a sketch style to a sheet layer viewport:

1. Select the viewport.
2. On the Shape tab of the Object Info palette, select Sketch from the **Rendering** list. Click **Render Settings** to select the viewport default sketch style.

   If you are creating a viewport, select Sketch from the **Rendering** list in the Create Viewport dialog box, and then click **Render Settings** to select the viewport default sketch style.

   The Sketch Render Settings dialog box opens. Select a sketch style for the viewport.

Click to show/hide the parameters.

Parameter	Description
Viewport Default Sketch Style	Specifies the sketch style to apply to the viewport
Edit	Opens the Sketch Style Editor dialog box, for changing the selected sketch style parameters
Preview	Temporarily applies the current sketch style to the viewport (disabled when accessed through the Create Viewport dialog box)

3. Click **OK**.

   The selected viewport sketch style overrides the document default sketch style for any objects within the viewport, including the crop object, referenced design layers, and any annotations.

   A sketch style cannot be directly applied to a viewport from the Render tab of the Object Info palette or by dragging the sketch style from the Resource Browser.

   A sketch style cannot be directly applied to a design layer viewport, but it can be applied to the layer of the design layer viewport.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Sketch Styles
Creating Sheet Layer Viewports

Sketching with Hidden Line Rendering

The default document sketch style can be applied to hidden line, dashed hidden line, and final shaded polygon rendering modes.

To apply a sketch style to hidden line rendering:

1. Select the default document sketch style as described in “Setting the Default Sketch Style” on page 1497.
2. Select **View > Rendering > Line Render Options**.

   The Line Render Options dialog box opens.

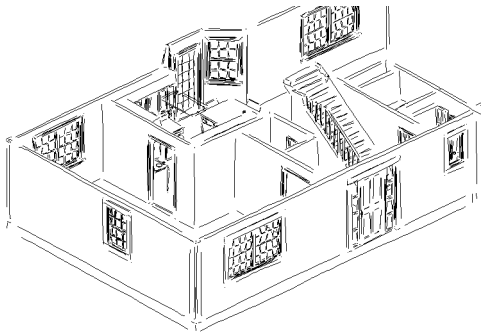
   Select **Sketch Hidden Line Results**, and select the sketch style from the list. Click **Edit** to edit the sketch style parameters. Then click **OK** to apply the sketch style to hidden line rendering.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. To sketch a viewport's hidden line rendering, select the viewport, and then choose a hidden line rendering mode from the **Rendering** list in the Object Info palette. Click **Render Settings** to access the associated hidden line render settings dialog box.

   The Hidden Line Render Settings dialog box opens.

4. Select **Sketch Hidden Line Results**, and select the sketch style from the list. Preview or edit the sketch style if desired.

5. The hidden line, dashed hidden line, or final shaded polygon rendering displays with the selected sketch style.



Setting the Default Sketch Style
Rendering with Vectorworks

Creating and Mapping Textures

Textures and Shaders

In the Renderworks product, textures are applied to 3D objects to make them appear more realistic. Textures are composed of components called shaders. There are four shader types: color, reflectivity, transparency, and bump; each shader makes an independent contribution to the overall texture appearance. When you create or edit textures, the shader components are combined or excluded to achieve the desired texture effect.

In addition to using textures, realistic models can be enhanced with the use of backgrounds, weather effects, and the use of image files as props.

Four shaders contribute to a texture definition. Combining and adjusting these components causes a texture to appear as an image, colored, bumpy, shiny, and/or transparent. The following table defines the shader types:

Shader	Function
Color	Defines surface color; this can be a plain, uniform color, or a complex pattern like wood or marble
Reflectivity	Defines amount of light reflected by surface; it is dependent on surface texture properties and any light sources
Transparency	Defines surface transparency or opacity
Bump	Defines surface irregularities which give the texture a bumpy appearance

> Textures can be associated with surface hatches to give the appearance of a texture in a monochromatic, hidden line rendering. See “Hidden Line Rendering with Surface Hatches” on page 1606.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Renderworks Shader Types

Creating Textures

Textures are created and displayed in the Resource Browser and are saved with the file (default content is automatically imported into the file at the point of use, and displays in the Resource Browser); see “Resource Libraries” on page 219 and “Using the Resource Browser” on page 221. Textures are created by defining the shaders that make up the texture.

Apply textures to an object from the Resource Browser or from the Render tab of the Object Info palette. See “Applying and Mapping Textures” on page 1527 for more information.

Textures can be associated with surface hatches to give the appearance of a texture in a monochromatic, hidden line rendering. See “Hidden Line Rendering with Surface Hatches” on page 1606.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a New Texture
Creating Image-based Shaders
Editing Textures and Shaders

Creating a New Texture

To create a texture resource:

1. Select **Window > Palettes > Resource Browser** to open the Resource Browser.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Renderworks Texture**.

The Edit Texture dialog box opens. Specify the shader parameters. Shader types and properties are described in “Renderworks Shader Types” on page 1546.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies the name of the texture resource
Shaders	Sets the type of shader or combination of shaders to use, and specifies the shader parameters.
Color	Color shaders apply a color or pattern to the surface of an object. Select a color shader from the list, or choose Object Attribute to apply the object’s fill color attribute. Image-based shaders require the selection of an image file. After selecting the shader, click **Edit** to edit the shader properties.
Reflectivity	Reflectivity shaders control how much light is reflected from a surface. Select a reflectivity shader from the list (or select None to exclude this type of shader from the texture). Image-based shaders require the selection of an image file. After selecting the shader, click **Edit** to edit the shader properties.
Transparency	Transparency shaders control surface transparency or opacity. Select a transparency shader from the list (or select None to exclude this type of shader from the texture). Image-based shaders require the selection of an image file. Mask-based transparency shaders create a transparent mask from an image based on specified settings. After selecting the shader, click **Edit** to edit the shader properties.
Bump	Bump shaders apply a texture with bumps or dimples to an object surface. Select a bump shader from the list (or select None to exclude this type of shader from the texture). Image-based shaders require the selection of an image file. After selecting the shader, click **Edit** to edit the shader properties.
Size	Sets the real-world size for each repetition of the texture If applying a brick shader, specify whether to adjust the brick dimensions proportionally to match the change to the texture size.
Set By Image	For image-based textures, opens the Set Image Size dialog box, to set the texture size using the image (see “Setting the Texture Size by Image” on page 1507)
Shadows	
Cast	Allows objects with this texture to cast shadows
Receive	Allows objects with this texture to receive shadows
Surface Hatch	Displays the name of the current surface hatch resource, for use in hidden line rendering with surface hatches. Surface hatches are an aspect of textures
Edit Surface Hatch	Opens the Edit Surface Hatch dialog box, to select the surface hatch to associate with the texture, and to align and register the surface hatch and texture (see “Editing Surface Hatches” on page 1606)
Preview Options	The Preview window displays the effects of shader and size selections on a preview object
Preview Controls	Adjusts the preview position and magnification. Click **Pan** and drag the preview to the desired location. Click **Zoom In** or **Zoom Out** and then click and drag to create a marquee; this zooms in or out on a particular section of the preview. Click **Fit** to fit the preview to the window (according to **Obj Size**).

Parameter	Description
Obj Type	Select the type of preview object from the list; for procedural (non image-based) shaders, the **Flat** object type is automatically used to create a preview for OpenGL rendering to approximate the look of the solid shader
Obj Size	Specifies the preview object size
Shader check boxes	Select to preview the associated shader; deselect to exclude the shader component from the texture preview
Indirect Lighting Options	Opens the Indirect Lighting Texture Options dialog box, to set any overrides for the texture when rendering with indirect lighting (see "Setting Indirect Lighting Options" on page 1507)
Revert	Returns the texture parameters to the original settings, undoing any changes

4. Click **OK** to create a texture with the name and properties specified in the Edit Texture dialog box. The texture displays in the Resource Browser and is saved with the file (the texture preview in the Resource Browser uses the **Flat** preview object at twice the texture size for ease of identification).

   If you are modeling a glass object (glass reflectivity shader) with a 3D polygon or other sheet-like 3D object, duplicate and offset the 3D polygon by a small amount so that rays are traced through the glass with both an entering and exiting surface.

Setting the Texture Size by Image

For image-based shaders, the real-world size of each texture repetition can be set based on a segment of the image.

To set the size of an image texture based on the image:

1. Select the image to use for the texture, as described in "Creating Image-based Shaders" on page 1508.
2. In the Edit Texture dialog box, click **Set By Image**. If several image-based shaders are used, select the shader with the desired image in the Choose Image dialog box.

   The Set Image Size dialog box opens. Red handles flash briefly to indicate the location of the line segment.
3. Specify the image length to use for sizing the texture by dragging the line segment into position, and then dragging the ends of the line segment. If necessary, use the mouse scroll wheel to zoom into and out of the image, or click and hold the mouse wheel button to pan.

   When the line is indicating the desired real-world length, specify the real-world size for the line segment in **Feature Size**.
4. Click **OK** to exit the Set Image Size dialog box and update the **Size** value.

Setting Indirect Lighting Options

To save rendering time, it is possible to override the indirect lighting settings for individual textures (see "Setting Lighting Options" on page 1571).

To set indirect lighting overrides for a texture:

1. From the Edit Texture dialog box, click **Indirect Lighting Options**.

   The Indirect Lighting Texture Options dialog box opens. Specify whether, and to what degree, the texture should override the indirect lighting options.

Click to show/hide the parameters.

Parameter	Description
Override Indirect Lighting	Sets whether the texture should participate in indirect lighting calculations.

Parameter	Description
Emit Light	Select this option if the texture should both receive and emit light
Receive Light	Select this option if the texture should receive light but not emit it

2. Click **OK**.

Creating Image-based Shaders
Renderworks Shader Types
Editing Textures and Shaders

Creating Image-based Shaders

Image-based shaders are created from image files imported into the Vectorworks file. Like wrapped shaders, an image-based shader is applied to the surface of an object. See “Applying and Mapping Textures” on page 1527 for more information. Different settings are then required depending on the type of shader to be used with the image.

Multiple image shaders can be combined to create a realistic texture. For example, import an asphalt image, and then add an image bump to the asphalt. In addition, image-based shaders can be combined with non image-based shader types for a variety of effects.

Most image-based textures are automatically compressed when imported. Imported JPG files retain the original JPG data; all other image files are compressed using lossless PNG format.

The Bricks shader includes its own image-selection functionality; see “Color Shaders” on page 1546.

Selecting the Image for Import

To import an image for use as an image-based shader:

1. Create a graphics file to be imported as a shader. In a graphics program, save the image in a format that the Vectorworks program can import:
   - BMP
   - JPG
   - PCT
   - PNG
   - PNT
   - PSD
   - QTI
   - SGI
   - TIF
   - TGA
2. Create a new texture as described in “Creating a New Texture” on page 1505. From any of the four shader component lists, select one of the image shader types.
3. If a resource with an image is already present in the file, the Choose Image dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Import an Image File	Imports a new image; click **OK** and proceed to Step 4.
Reuse an Image from Another Resource	Reuses a previously imported image; select the resource or other shader type that contains the image. Click **OK** and proceed to Step 5.

4. Select the desired image file in the Import Image Document dialog box. Click **Open**.
5. The next dialog box that opens depends on the type of shader. Refer to the section that applies to the shader.

Shader	Section
Color	“Importing Images for Color Shaders” on page 1509
Reflectivity	“Importing Images for Reflectivity Shaders” on page 1510
Transparency	“Importing Images for Transparency Shaders” on page 1511
Bump	“Importing Images for Bump Shaders” on page 1514

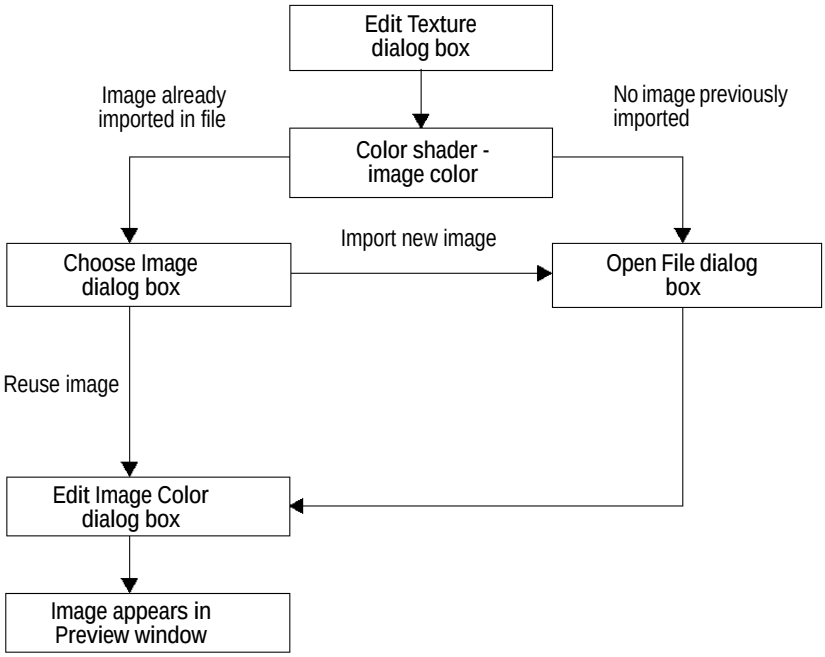
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a New Texture
Editing Textures and Shaders

Importing Images for Color Shaders

Color shaders can use images for image color and as a filtered image. The image color shader can display the image as-is, or can tint the image with a specified color.

The process of selecting a color image shader is illustrated by the following flow chart.



To import a color image shader:

1. Select and import the image as described in “Selecting the Image for Import” on page 1508.

   The Edit Image Color dialog box opens. Specify the image-based shader properties.

Click to show/hide the parameters.

Parameter	Description
Image preview	Edits to the image are displayed in the image preview
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

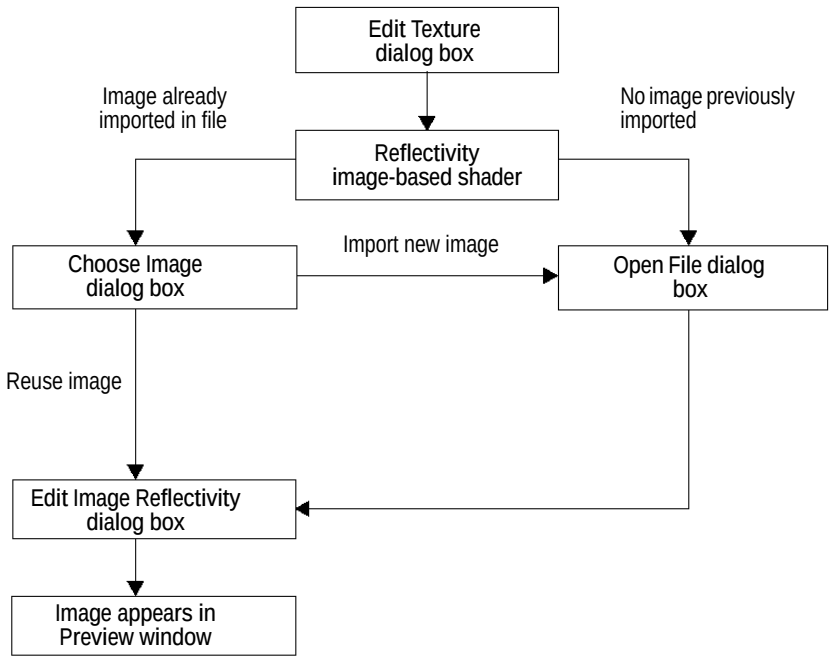
Parameter	Description
Tile Image	Repeats the image in the horizontal, vertical, or horizontal and vertical directions; deselect for no tiling
Filter Color	
No filter	Does not change the image with a filter color
Use Object Fill	Filters the color of the image by the fill color selected for the object (can be different for each object that is textured by this texture)
Use Chosen Color	Tints the image with the selected color; click the color box to choose a filter color for the image
Change Image	Selects a different image for import
Flip H/V	Flips the image horizontally or vertically
Rotate	Rotates the image 90° counter-clockwise
Invert	Produces a negative of the image

2. Click **OK** to import the image and preview the texture.

Applying Colors
Creating Image-based Shaders
Renderworks Shader Types
Editing Textures and Shaders

Importing Images for Reflectivity Shaders

The process of selecting a reflectivity shader is illustrated by the following flow chart.



To import a reflectivity image-based shader:

1. Select and import the image as described in “Selecting the Image for Import” on page 1508.

The Edit Image Reflectivity dialog box opens. Specify the image-based shader properties.

Click to show/hide the parameters.

Parameter	Description
Image preview	Edits to the image are displayed in the image preview
Change Image	Selects a different image for import
Flip H/V	Flips the image horizontally or vertically
Rotate	Rotates the image 90° counter-clockwise
Invert	Produces a negative of the image
Reflection (%)	Sets the amount of reflection. Generally, white pixels are the most reflective, while colored pixels reflect in their color.
Blurriness (%)	Sets how blurry the reflection appears. A range of 0 – 40% is typical.

2. Click **OK** to import the image and preview the texture.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

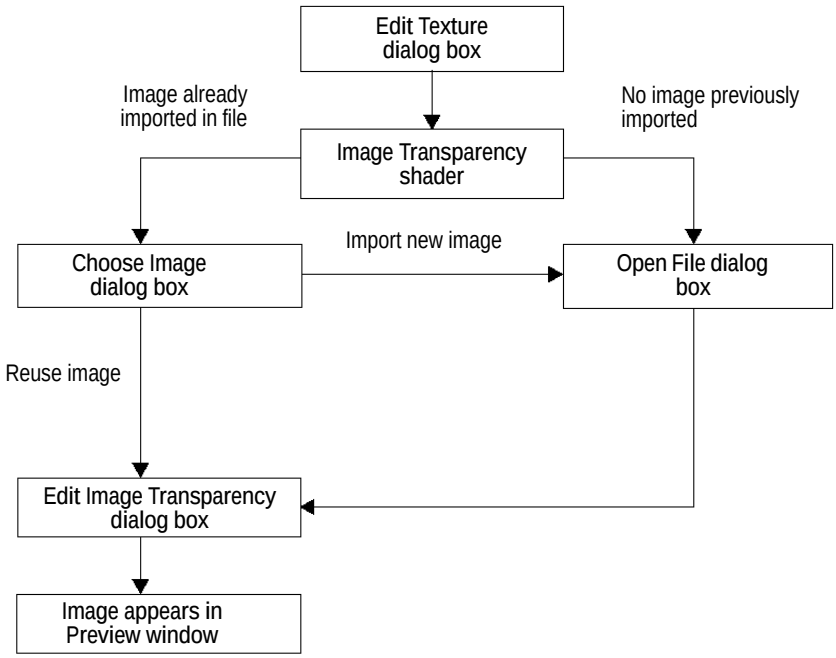
Creating Image-based Shaders
Renderworks Shader Types
Editing Textures and Shaders

Importing Images for Transparency Shaders

Transparency shaders can be created from images or image masks.

Image Transparency

The process of selecting an image transparency shader is illustrated by the following flow chart.



To import an image transparency shader:

1. Select and import the image as described in “Selecting the Image for Import” on page 1508.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Edit Image Transparency dialog box opens. Specify the image-based shader properties.

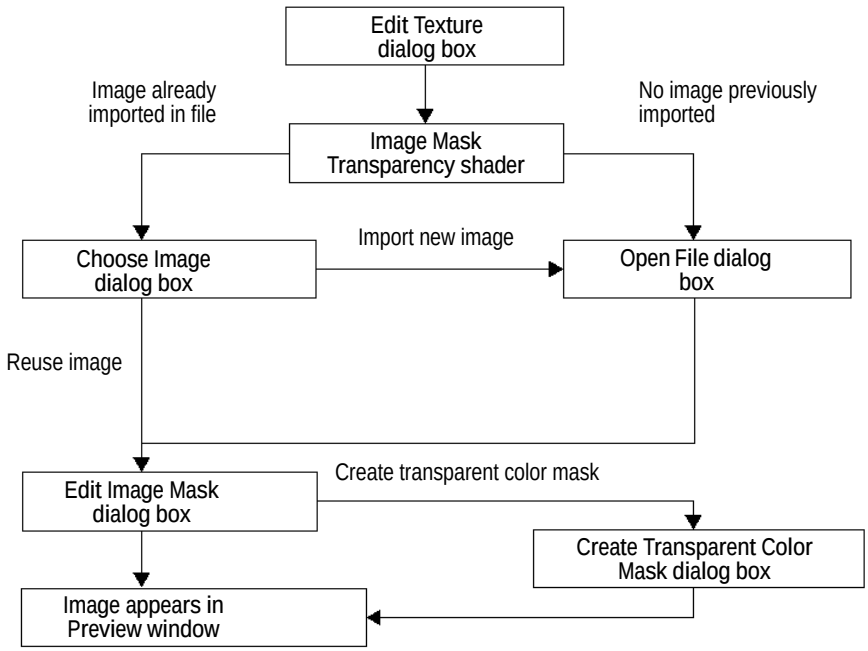
Click to show/hide the parameters.

Parameter	Description
Image preview	Edits to the image are displayed in the image preview
Index of Refraction	As light moves through a medium, the index measures the change in the direction of the light's rays. An index of 1.0 indicates none; a typical value for water and ice is 1.3, and for glass use 1.5 – 1.6.
Blurriness (%)	Sets image blurring when rendered in Final Quality or Custom Renderworks (with **Blurriness** selected)
Absorption Color	Click the color box to select a color that the object may absorb differently, making it appear tinted by that color
Absorption Distance	Defines the distance rays of light have to travel before **Absorption Color** replaces the image color. The lower the value, the more intense the **Absorption Color** is.
Change Image	Selects a different image for import
Flip H/V	Flips the image horizontally or vertically
Rotate	Rotates the image 90° counter-clockwise
Invert	Produces a negative of the image

2. Click **OK** to import the image and preview the texture.See "Transparency Shaders" on page 1564 for more details about transparency parameters.

Mask Transparency

The process of selecting a mask transparency shader is illustrated by the following flow chart.



To create a mask transparency image-based shader:

1. Select and import the image as described in "Selecting the Image for Import" on page 1508.

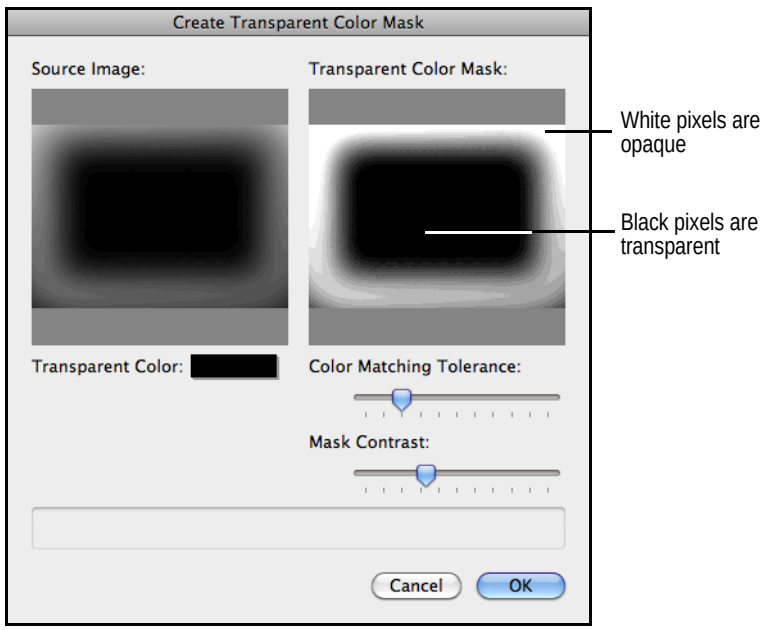
2. The Edit Image Mask dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Horizontal/ Vertical Repetitions	Specifies the number of repetitions of the mask shader in the horizontal and vertical direction
Single	The mask shader does not repeat; it is only shown once (this is the typical setting, especially for decals)
Infinite	The mask shader repeats infinitely in the horizontal and/or vertical direction
Custom	The mask shader repeats the specified number of times
Anti-Aliased	Select to smooth the edges of the image mask
Change Image	Selects a different image for import
Transparent Color Mask	Edits the transparent color mask settings
Flip H/V	Flips the image horizontally or vertically
Rotate	Rotates the image 90° counter-clockwise
Invert	Produces a negative of the image; useful for swapping the opaque and transparent areas of a grayscale mask

3. To use the image as a mask, click **Transparent Color Mask**; the Create Transparent Color Mask dialog box opens. Select the mask properties.

   The source image must be more than eight-bit color to create a transparent color mask. Images with a monochrome background are easiest to use when creating a mask transparency.



Click to show/hide the parameters.

Parameter	Description
Source Image	Displays the imported image. Select the transparent color by clicking a color in the image; the resulting mask is displayed in the Transparent Color Mask preview. If necessary, use the mouse scroll wheel to zoom into and out of the image, or click and hold the mouse wheel button to pan.
Transparent Color	Displays the current transparent color. Either click on the source image to designate the transparent color, or select the color by clicking the color box.
Transparent Color Mask	Displays a preview of the mask based on the current transparent color selection and settings
Color Matching Tolerance	Adjusts the transparency tolerance; drag the slider to the right to increase the tolerance level. This allows a wider range of pixels similar to the transparent color to be considered transparent.
Mask Contrast	Adjusts the mask edge contrast; increase the edge contrast sharpness by dragging the slider to the right. Soften the contrast by dragging the slider to the left.

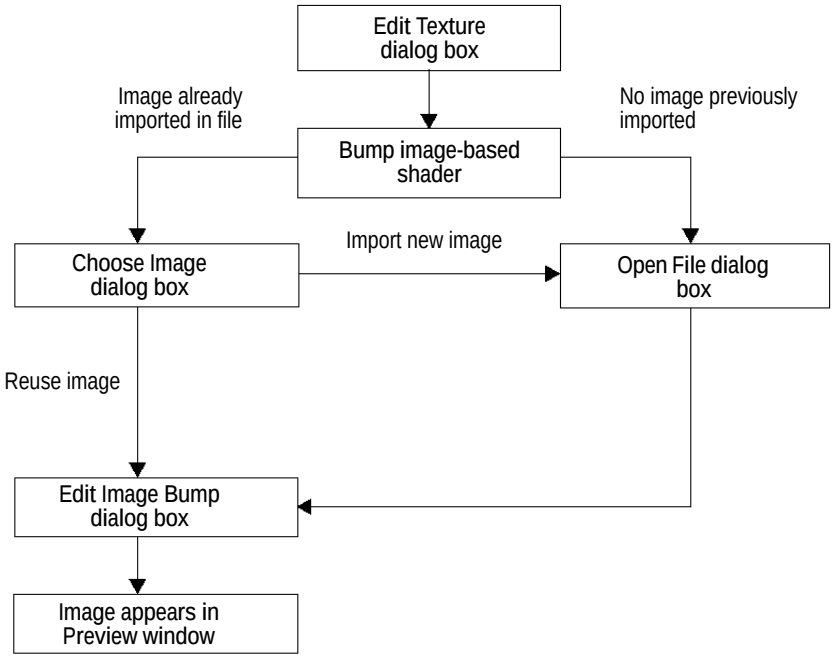
4. Click **OK**. The mask is previewed in the Edit Image Mask dialog box.
5. Click **OK** again. The imported mask transparency is previewed in the Edit Texture dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Image-based Shaders
Renderworks Shader Types
Editing Textures and Shaders

Importing Images for Bump Shaders

The process of selecting a bump shader is illustrated by the following flow chart.

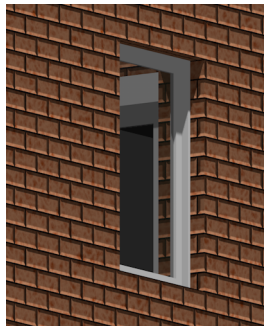
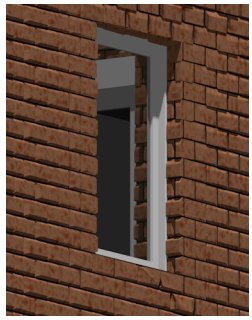


To import a bump image-based shader:

1. Select and import the image as described in “Selecting the Image for Import” on page 1508.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Edit Image Bump dialog box opens. Specify the image-based shader properties.

Click to show/hide the parameters.

Parameter	Description
Image preview	Edits to the image are displayed in the image preview
Change Image	Selects a different image for import
Flip H/V	Flips the image horizontally or vertically
Rotate	Rotates the image 90° counter-clockwise
Invert	Produces a negative of the image; useful for reversing the high and low pixels for a bump image
Bump Strength (%)	Sets the amplitude of the bump shader. Positive and negative values can be entered. A value of 10% would result in a mild bumpy appearance.
Displacement Mapping	For realistic bump textures, displacement mapping creates texture and bumpy details with a rendering technique that appears embossed, projecting the geometry outward from the surface.  Bump shader without displacement mapping  Bump shader with displacement mapping This mapping applies only to Final Quality and Custom Renderworks, when **Displacement Mapping** is enabled in the render options. Rendering can be significantly slower with displacement mapping. If the image bump does not provide the desired results, try a noise bump shader.
Height	Specify a non-zero height to enable displacement mapping; large height values may result in longer render times
Detail	Sets the level of detail for displacement mapping; requirements and results vary depending on the texture and the surface's face size. Textures without too much bump detail and a large face size, such as boards or stones, render with less detail and can be set with a lower level of detail; fine, faceted textures like grass or leaves may require a high level of detail, which also requires more rendering time. Conversely, very large surfaces, like a ground plane, may need higher levels of detail to see the displacement.
Self-Shadowing	Adds shadows to the displaced geometry, increasing realism as well as rendering time

2. Click **OK** to import the image and preview the texture.

When you use the monochromatic textures provided with the Vectorworks program, the **Bump Strength** parameter may require careful adjustment depending on the viewing distance from the model.

Due to compression artifacts, JPG images generally do not work well as bump images. PNG and TIF compression works well for bump images; when importing images, select PNG rather than JPG as the compression method.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Image-based Shaders
Renderworks Shader Types
Editing Textures and Shaders

Creating a Texture from a Hatch

Textures can be created from existing hatches to create an associated texture and, optionally, surface hatch resource for use in rendering. The resulting image-based texture is created automatically from the hatch. The texture's size is based on the calculated repeat interval of the hatch.

To create a texture from a hatch:

1. Right-click (Windows) or Ctrl-click (Mac) on the hatch in the Resource Browser and select **Create Texture from Hatch** from the context menu.
2. Name the texture and click **OK**.

Editing Textures and Shaders

All textures, whether created by you, selected from default content, or imported from the resources included with the Vectorworks program, can be edited by changing the parameters of the shaders that make up the texture. This editing can be done from the Object Info palette, with the changes to the texture reflected immediately in the selected object. Alternatively, the texture can be edited from the Resource Browser.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing Arroway Textures
Editing Textures and Shaders of Selected Objects
Editing Textures and Shaders Through the Resource Browser

Importing Arroway Textures

Arroway textures™ are high-quality, high-resolution, image-based textures available from www.arroway-textures.com. In the Renderworks product, low-resolution image-based textures from Arroway are available as default content. Default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see "Resource Libraries" on page 219. Arroway textures are easily identified because they contain "Arroway" in the texture name.

If your file contains low-resolution Arroway textures and you want to update them to their high-resolution equivalents, or if you have purchased high-resolution Arroway textures and would like to use them in a Vectorworks file, the **Import Arroway Textures** command accomplishes this automatically. This command is not available by default, but must be added to the workspace.

To update low-resolution Arroway textures or import new Arroway textures:

1. Purchase the required high-resolution textures from Arroway and place them in a folder. Organize the purchased textures so that the ones to be imported or updated are contained in a folder, and optionally in subfolders.
2. Add the **Import Arroway Textures** command to a new or existing workspace (see "Modifying Menus and Commands" on page 1837). The command can be found among the Import/Export items on the Menus tab of the Workspace Editor.
3. Select **Import Arroway Textures** from the custom location in the workspace.

The Import Arroway Textures dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Create New Textures	Creates new image-based textures from the Arroway high-resolution textures in the specified folder(s)
Update Existing Textures	Updates any low-resolution Arroway textures in the current file with their high-resolution equivalent
Max Pixel Dimension	Specifies the maximum dimension for the imported textures; choose the lowest practical pixel dimension to avoid memory issues, particularly when several textures are used

4. Set the parameters and click **Import**.

   The Import Arroway Textures Folder dialog box opens.

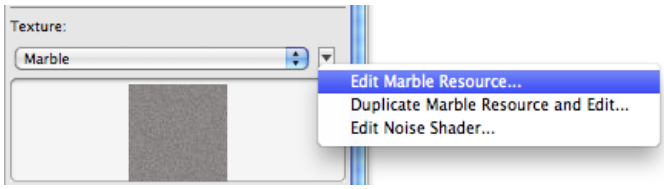
5. Select the folder containing the high-resolution Arroway textures you have purchased.

   Using several very large images may affect performance.

6. Click **Select Folder** to import or upgrade the Arroway textures from the selected folder and any subfolders. Renderworks textures are automatically created or updated in the current file.

Editing Textures and Shaders of Selected Objects

The textures and shaders of a selected object can be edited directly through the **Texture** menu on the Render tab of the Object Info palette. The **Texture** menu allows the textures and shaders that make up the textures to be edited, with changes displayed immediately in the drawing. The texture mapping can also be edited from the Object Info palette; see “Applying a Texture to an Object” on page 1530.



To avoid making unwanted permanent changes to a texture resource that is used by several objects, select **Duplicate (Texture Name) Resource and Edit** and work on a copy of the texture resource instead of the original. Changes made to a copy of the texture only apply to the current selection.

Editing the Texture Resource

To edit the texture resource of a selection:

1. Select the desired textured object(s).
2. On the Render tab of the Object Info palette, click the arrow next to the texture name to open the **Texture** menu.
3. Select **Edit (Texture Name) Resource** to edit the current texture, or select **Duplicate (Texture Name) Resource and Edit** to make a copy of the resource for editing.

   The Edit (Texture Name) dialog box opens, and a preview window is created around the selected object.

   The parameters are the same as those used to create the texture; any changes are reflected directly in the drawing.

   Some parameters are not available when editing a resource from the Object Info palette; to edit those parameters you must access the texture directly from the Resource Browser.

Click to show/hide the parameters.

Parameter	Description
Shaders	Select a different or additional shader, or edit the shader parameters, as described in “Creating Textures” on page 1505
Size	Sets the real-world size for each repetition of the texture
Set By Image	For image-based shaders, click to set the real-world size in relation to a segment of the image (if several image-based shaders are used, select the shader in the Choose Image dialog box first). In the Set Image Size dialog box, specify the texture length to use for sizing by dragging the ends of the line segment. Then, specify the real-world size for the line segment in **Feature Size**. Click **OK** to exit the Set Image Size dialog box and update the **Size** value. The size should reflect the file setup. For example, if the layer scale is 1/4” and feet and inches are being used, the size should be in feet, not inches.
Shadows	
Cast	Allows objects with this texture to cast shadows (for rendering modes that display shadows)
Receive	Allows objects with this texture to receive shadows (for rendering modes that display shadows)
Indirect Lighting Options	Opens the Indirect Lighting Texture Options dialog box, to set any overrides for the texture when rendering with indirect lighting (see “Setting Indirect Lighting Options” on page 1507).
Preview Options	
Render Mode	Select a rendering mode for the preview window; this does not change the drawing rendering mode
Update When Values Change	Select to render the preview with the selected **Render Mode** as parameters change; if deselected, the drawing is not updated until the dialog box is closed
Render Selected Object Only	Select to render only the selected object as parameters change; deselect to render objects within the preview window as parameters change
(Texture Name) is Used by	Indicates the number of objects in the file with the current texture applied

Editing the Shader

If only the shader parameters require editing, they can be accessed quickly from the **Texture** menu rather than editing the texture first.

To edit one of the shaders that makes up the texture of a selection:

1. Select the desired textured object(s).
2. On the Render tab of the Object Info palette, click the arrow next to the texture name to open the **Texture** menu.
3. Select **Edit (Shader Name) Shader** to directly edit one of the shaders that makes up the applied texture.

   The Edit (Shader Name) Shader dialog box opens, and a preview window is created around the selected object.

   The parameters are the same as those used to create the shader; any changes are reflected directly in the drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Renderworks Shader Types
Editing Textures and Shaders Through the Resource Browser
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

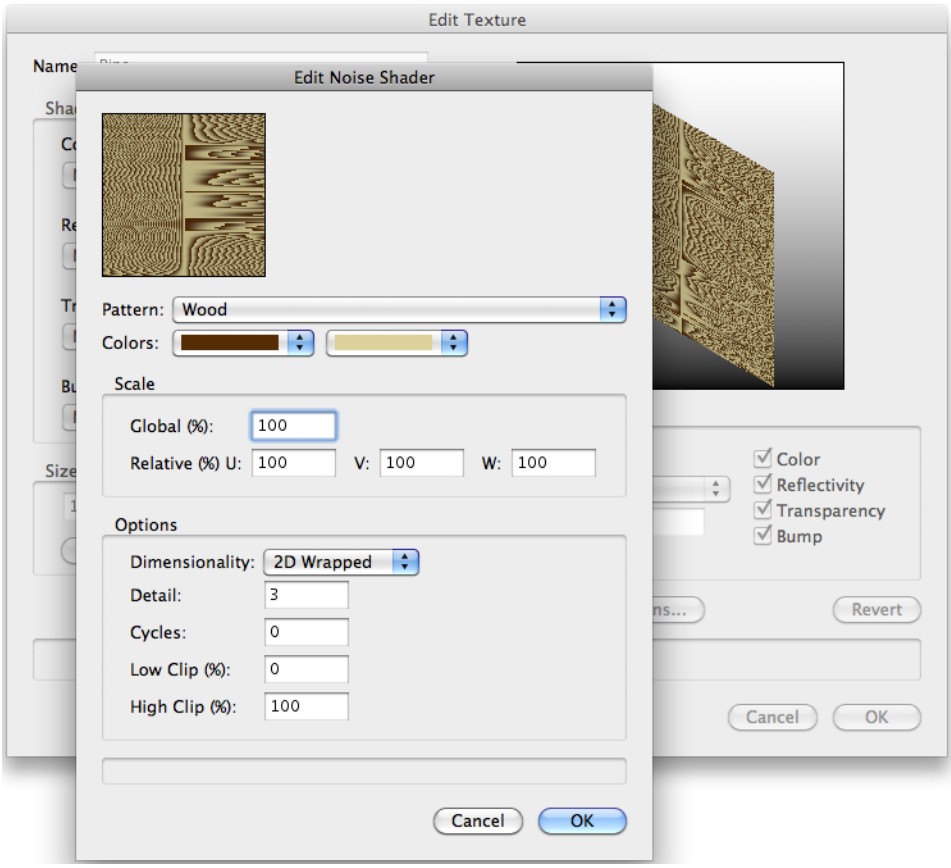
Editing Textures and Shaders Through the Resource Browser

A texture and its shader properties can be edited from the Resource Browser. The texture does not have to be applied to any objects to be edited in this way; changes are displayed in a preview window rather than directly in the drawing.

To edit a texture resource, including its shaders, from the Resource Browser:

1. From the **Resource Browser**, select the texture to be changed, and then select **Resources > Edit**.

   The Edit Texture dialog box opens.



2. Click **Edit** to edit one of the shaders that make up the texture. The dialog box that opens depends on the shader. Descriptions of all the possible parameters are presented in alphabetical order in “Renderworks Shader Types” on page 1546.
3. Click **OK** to exit the shader properties.
4. Edit the texture properties as described in “Creating Textures” on page 1505. The texture can also be renamed.

   The changes are displayed in the preview window.
5. Click **OK** to exit the Edit Texture dialog box. Textures applied to any objects are updated to reflect the changes.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Textures and Shaders of Selected Objects
Creating Textures
Renderworks Shader Types
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Image Prop Objects

The Renderworks product allows the insertion of "prop" objects made from imported images, such as trees, people, signs, and cars, to make a model look more realistic. An image prop must be in one of the formats listed in "Selecting the Image for Import" on page 1508. Most image-based textures are automatically compressed when imported into a Vectorworks file. Imported JPG files retain the original JPG data; all other image files are compressed using lossless PNG format.

For proper display when rendered, an image prop should have a fill style other than None in the Attributes palette.

An image imported for use as a prop is saved as a texture resource. A library of image prop resources is provided in the image props folder of [Vectorworks]\Libraries\ (see "Resource Libraries" on page 219).

To add an image prop:

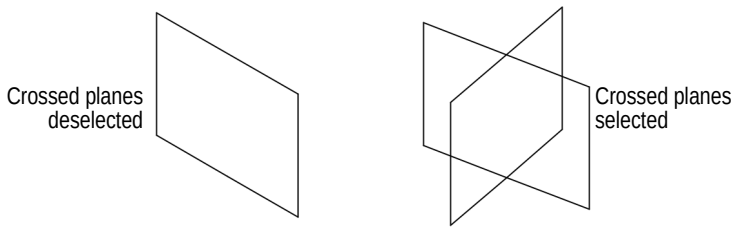
1. Select **Model > Create Image Prop**.
2. If a resource with an image is already present in the file, the Choose Prop Image dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Import an Image File	Imports a new image; click **OK** and proceed to Step 3
Reuse an Image from Another Resource	Reuses a previously imported image; select the resource that contains the image. Click **OK** and proceed to Step 4.

3. Select the desired image file in the Import Image Document dialog box. Click **Open**.
4. The Image Prop Options dialog box opens. Specify the parameters for the image prop.

Click to show/hide the parameters.

Parameter	Description
Name	Specify a name for the image prop (used for the texture and symbol name)
Dimensions	Specifies the image prop's height and width values
Lock Aspect Ratio	Maintains the image's aspect ratio when selected; editing one dimension automatically changes the other
Mask Options	
No Mask	Specifies that no image masking is to be used for the imported image
Use Mask	Specifies that image masking is to be used; click **Create Mask** to specify the mask image and properties as described in "Mask Transparency" on page 1512
Crossed Planes	Creates the illusion of object fullness by creating two image planes at a 90 degree angle to each other 

Parameter	Description
Constant Reflectivity	Excludes the prop from the effects of light objects placed in the drawing; this is useful for images that already contain shadowed areas
Create Plug-In Object	Creates a plug-in object out of the prop; this allows the prop to be resized and automatically rotated
Auto Rotate to Viewer	Adjusts the plug-in object so that it is always rendered facing the viewer
Create Symbol	Creates a symbol from the image prop; the image prop name is automatically assigned as the symbol name. Generating the image prop as a symbol allows the image prop to be easily re-inserted in the drawing.

Select **Auto Rotate to Viewer** for crossed plane images, so that the crossed planes cannot be detected.

5. Click **OK** to create the image prop.

   The image prop symbol parameters can be edited in the Object Info palette. In the Resource Browser, the image prop is listed as a symbol/plug-in object, if selected at creation. In addition, a texture is created for use by the image prop. Both the texture and, if created, the symbol/plug-in object, are assigned the name provided in the Image Prop Options dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding 3D Plants

Adding 3D Plants

Many designs have a need for plants that render quickly and appear more realistic than image prop objects. The Renderworks product includes several high-quality 3D plants from VBvisual, makers of 3D content for rendering packages. These plants look authentic in all views and cast realistic shadows, without adding excessively to rendering times or file size. The Renderworks product includes three free plants; more plants can be purchased to add to the library of 3D plants.

To place a 3D plant:

1. Click the **VBvisual Plant** tool from the Visualization tool set.
2. Select the plant to insert from the VBvisual Plant list on the Tool bar.
3. Click to insert the selected plant into the drawing.

   When applicable, the low resolution, summer version of the plant is inserted by default. The plant includes a 3D locus so it can be easily moved; in addition, the plant can be sent to the surface of a site model (see “Sending Objects to the Site Model Surface” on page 705). In plan view, a 2D version of the plant displays.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The object properties can be edited from the Object Info palette. The options available depend on the selected plant type.

Click to show/hide the parameters.

Parameter	Description
Name	Select the plant species from the list of included or purchased plants, or click **More Plants** to purchase additional plants
Height	Select the plant height from the list of available heights for the selected plant
Resolution	Certain plants include both a low- and high-resolution version. Selecting a high resolution provides the best appearance, but takes longer to render.
Season	Select the season for the plant to display

More plants are available for purchase in Vectorworks file format from VBvisual. Purchased plants must be placed in the default content library folder named VBvisual Plant (see “Resource Libraries” on page 219).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Image Prop Objects

Creating Layer Backgrounds

With the Renderworks product, backgrounds can be added to create a more realistic drawing. Backgrounds, such as clouds, a color or color gradient, or an imported image or panorama, are applied on a per-layer basis. A physical sky background can be combined with a heliodon (Vectorworks Design Series required) to create a realistic sky, correct in appearance for the time of day. In addition, a special lit fog effect can add the illusion of shafts of light in fog (point or spot light required).

Renderworks backgrounds are resources that are created and displayed in the Resource Browser and are saved with the file (default content is automatically imported into the file at the point of use, and displays in the Resource Browser); see “Resource Libraries” on page 219. Several Renderworks styles make use of backgrounds.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Background
Applying Renderworks Backgrounds

Creating a Background

To create a background:

1. Select **Window > Palettes > Resource Browser** to open the Resource Browser.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Renderworks Background**.

   The Edit Renderworks Background dialog box opens.
4. Enter the name of the new background and select the type of background to create. Click **Options** to set specific background parameters.

Click to show/hide the parameters.

Parameter	Description
None	Select this option when creating a lit fog effect without an additional background

Parameter	Description
Clouds	Specifies a sky background with a background color and clouds
Scale	Specifies the cloud scale (1 – 10); enter a larger value for larger cloud sizes
Background Color	Select a background color for the sky
Cloud Color	Select a cloud color
Detail	Specifies the level of cloud resolution; drag the slider to the right to specify a greater level of detail (requires longer rendering times)
One Color	Specifies a background with one color; click **Options** to select the color
Two Color	Specifies a background with two colors which blend to form a gradient; click **Options** to select the colors
Top Color	Select the color for the start of the color gradient at the top of the page
Bottom Color	Select the color for the end of the color gradient at the bottom of the page
Image	Uses a specified image for the background (see "Creating Image Backgrounds" on page 1524)
Image Environment (HDRI)	Uses a panoramic High Dynamic Range Image (HDRI) file for the background (see "Creating HDRI Backgrounds" on page 1524)
Physical Sky	Uses environment lighting to realistically simulate various sky conditions (see "Creating Physical Sky Backgrounds" on page 1526)
Lit Fog	Creates volumetric lighting effects, such as a shaft or cone of scattered light in fog, haze, or smoke. A point or spot light object must be inserted in the drawing, and **Lit Fog** must be selected in the Object Info palette for the light source object. Lit fog cannot be applied to directional, area, or line lights.
Fog Density (%)	Specifies the fog density; enter a larger value to increase the density of the effect
Fog Consistency	Creates either a smooth or uneven fog effect
Receive Shadows	Allows the effect to be blocked by solid objects, creating shadows; for example, when blocking a shaft of light by a table, so that light does not display under the table **Use Shadows** must also be selected in the rendering options.

OpenGL render mode only displays One Color, Two Color, and Image backgrounds.

Click here for a video tip about this topic (Internet access required).

5. Click **OK** from the Create Renderworks Background dialog box to create the background resource.

   The new background resource is listed in the Resource Browser. It is available in the Edit Design Layers dialog box (see "Applying Renderworks Backgrounds" on page 1545 for information).

   A Renderworks background can be quickly edited by pressing Ctrl (Windows) or Option (Mac) and double-clicking on the resource in the Resource Browser.

   Backgrounds and lit fog effects can be used together.

   Textures and backgrounds that are not used in the drawing file should be purged to reduce file size. See "Purging Items from a File" on page 1011.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Renderworks Backgrounds
Setting Lighting Options
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Colors

Creating Image Backgrounds

To be used for a background, an image must be in one of the formats listed in “Selecting the Image for Import” on page 1508.

To create an image background:

1. Create the background resource as described in “Creating Layer Backgrounds” on page 1522.
2. Select Image from the **Background** list.
3. Select a new image or an image from either the default content or the current file’s content (see “Resource Libraries” on page 219).

   The Edit Image Background dialog box opens; specify the image height and width (centered on the page). An image used as a background of this type, as opposed to the HDRI background, remains fixed to the page and does not change according to the 3D view.

   An image can also be reused from another image-based resource if one exists in the file. In the Choose Image dialog box, select **Reuse an Image from Another Resource** and specify the resource. The Edit Image Background dialog box opens to allow editing of image parameters (see “Selecting the Image for Import” on page 1508 for more information).

Click to show/hide the parameters.

Parameter	Description
Width	Sets the width of the image in page units (inches or millimeters); width edits also automatically change the height, to maintain the image aspect ratio
Height	Sets the height of the image in page units (inches or millimeters); height edits also automatically change the width, to maintain the image aspect ratio
Change Image	Selects a different image for import
Flip H/V	Flips the image horizontally or vertically
Rotate	Rotates the image 90° counter-clockwise
Invert	Produces a negative of the image

4. Click **OK** to use the image in the background.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Renderworks Backgrounds
Creating a Background
Setting Lighting Options
Applying Colors

Creating HDRI Backgrounds

When an HDRI background is imported, it behaves as if an infinitely large, textured, sphere or cube was centered on the model. The virtual sphere or cube is so large that view position changes do not affect its display. However, view orientation changes do affect which part of the image is shown in the background; this provides a true 3D sense to the model. One bounce of indirect lighting is generated with HDRI lighting.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

By default, an image environment background renders as both a background and a light source. However, it is possible to use one HDRI background resource as an environment background, and another for environment lighting; see “Setting Lighting Options” on page 1571. Several Renderworks styles incorporate the use of an HDRI background.

Purpose of HDRI Background Resource	Creation Method
Use an HDRI background resource as both environment background and environment lighting	Create an HDRI background resource and apply it to the layer. By default, the Layer Lighting Options is set to use the lighting from the current background.
Use an HDRI background as environment background only	Create an HDRI background resource and apply it to the layer. In the Layer Lighting Options dialog box, specify **None** for Environment Lighting.
Use an HDRI background resource as environment lighting only	Create an HDRI background resource, but do not apply it to the layer. In the Layer Lighting Options dialog box, select **From Selected Background** and specify the resource.
Use one HDRI background resource as an environment background and a different background resource for environment lighting	Create the first HDRI background resource and apply it to the layer (environment background). Create the second background resource but do not apply it to the layer. Instead, in the Layer Lighting Options dialog box, select **From Selected Background** and specify the second resource (environment lighting).

HDRI backgrounds render best in Perspective projection. In Orthogonal projection, HDRI backgrounds are suitable for lighting and reflections, but appear as a single background color because of the narrow field of view.

If the HDRI background includes a sun in its image, and the drawing also contains directional lights, the light directions may not match, creating multiple shadows.

To create an HDRI background:

1. Create the background resource as described in “Creating Layer Backgrounds” on page 1522.
2. Select Image Environment (HDRI) from the **Background** list.
3. Select the .hdr image file to use. The image is automatically converted to Lat/Long format. An OpenEXR file can also be used; it must be in Lat/Long format.

   The Edit Image Environment (HDRI) Background dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Preview	Displays a preview of the selected image

Parameter	Description
Rotation	Rotates the image clockwise about the Z axis, changing the light orientation and visible portion of the image (*range*: -180 to 180 degrees; positive angles are rotated clockwise)
Change Image	Imports a different image to use as the HDRI background
Brightness (%)	Specifies the brightness for environment lighting; enter a percentage or drag the slider to change the brightness. A value over 100% can be entered.
Saturation (%)	Specifies the color saturation for environment lighting; enter a percentage or drag the slider to change the saturation. A value over 100% can be entered.
Quality	Specifies the sampling value of the rendered image; higher-quality sampling results in better image resolution, but longer rendering times. Select From Render Mode to use the **Soft Shadows** quality specified in the render options (see "Custom Renderworks Options" on page 1600).

4. Set the parameters and click **OK** to use the image as an environment background resource.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Renderworks Backgrounds
Creating a Background
Creating Physical Sky Backgrounds
Setting Lighting Options
Applying Colors
Renderworks Styles

Creating Physical Sky Backgrounds

A physical sky creates the realistic appearance of a sky background with various levels of cloudiness. When used in conjunction with the heliodon (Vectorworks Design Series required), the sky's appearance is appropriate for the set time of day and can change its appearance automatically during a solar animation. Several Renderworks styles incorporate the use of a physical sky background; these are preset to give the design a specific look at a certain time of day. However, you can also create your own physical sky background for use in rendering.

Because the physical sky uses environment lighting (see "Creating HDRI Backgrounds" on page 1524), the sky provides a light source.

To create a physical sky background:

1. Create the background resource as described in "Creating Layer Backgrounds" on page 1522.
2. Select Physical Sky from the **Background** list.

   The Edit Physical Sky Background dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Preview	Displays a preview of the sky condition
Sky Condition	Select the amount of cloud cover for the sky background
Brightness (%)	Specifies the brightness of the sky background; enter a percentage or drag the slider to change the brightness. A value over 100% can be entered.
Saturation (%)	Specifies the color saturation of the sky background; enter a percentage or drag the slider to change the saturation. A value over 100% can be entered.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Set the parameters and click **OK** to use the physical sky in the background.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Renderworks Backgrounds
Creating a Background
Creating HDRI Backgrounds
Setting Lighting Options
Solar Studies
Renderworks Styles

Applying and Mapping Textures

Once a texture resource has been created, it can be applied either to a selected object or to objects assigned to a class. Mapping specifies how a texture is applied to an object; different types of objects require different mapping types.

> 2D objects cannot have a texture applied; in this case, the Render tab of the Object Info palette displays the message "Cannot apply texture to object."

When you select a textureable object, the Render tab of the Object Info palette displays controls to assign a texture and adjust the texture mapping to achieve various visual effects. The result of these parameter changes is visible immediately in the drawing, as the selected object is re-rendered. Some objects have specific parts, which can have different textures and mapping types; for example, the texture and mapping type applied to the Top part might be different from those applied to the Sides. For some mapping types, you can then use the **Attribute Mapping** tool to adjust the texture mapping directly.

Decals are special textures that are applied over other textures. They are created from imported images or existing texture resources.

Finally, a Renderworks background can be applied to add impact and realism to the design.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Texture Projection and Orientation
Applying a Texture to an Object
Direct Texture Mapping
Creating Decal Textures
Applying Textures to Symbols, Walls, and Roofs
Applying Renderworks Backgrounds

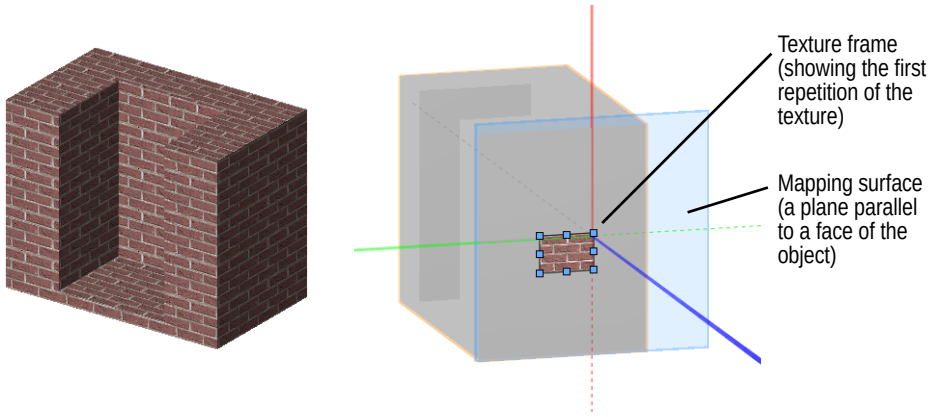
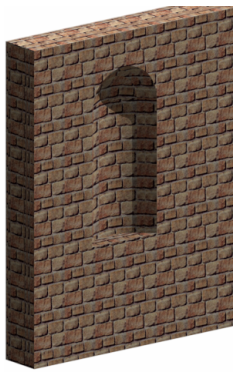
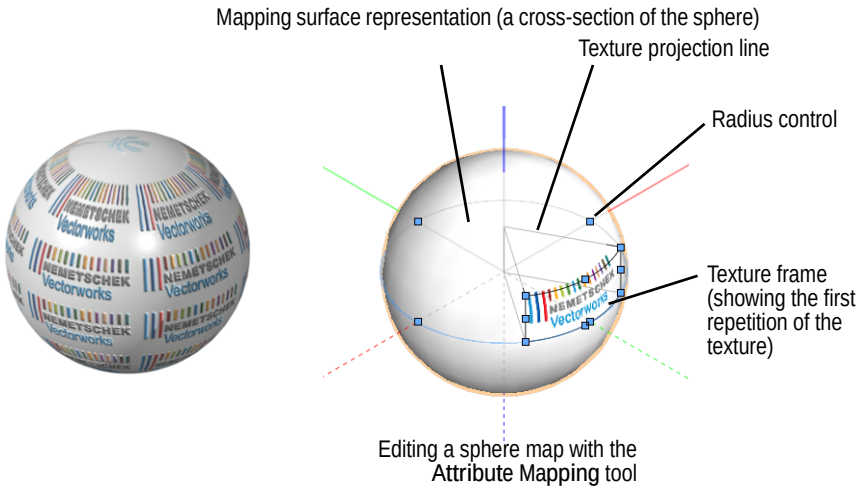
Texture Projection and Orientation

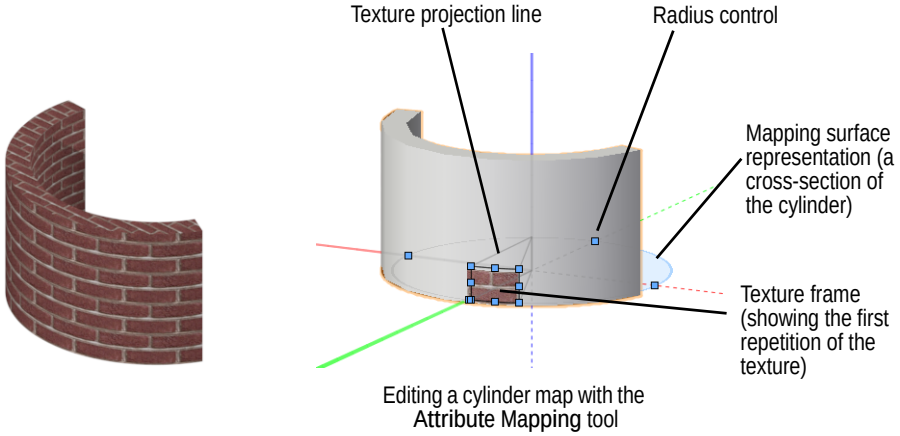
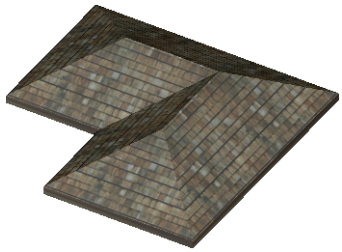
Map Types

A map type is used to control how a texture is projected onto different types of objects in a general way. Plane, auto-align plane, sphere, cylinder, perimeter, roof, and imported mapping types are available. To further adjust the mapping, the projected texture can be rotated, moved, and scaled using the Render tab of the Object Info palette. For plane, sphere, and cylinder mapping types, you can use the **Attribute Mapping** tool to adjust the mapping directly on the object.

> Surface hatches observe the scaling, rotation, and offset applied to the defining texture.
>
> Surface hatches associated with a texture support Plane, Auto-Align Plane, and Cylinder texture map types. Sphere, Perimeter, and Roof texture map types are rendered as Auto-Align plane for surface hatches. Surface hatches on some curved surfaces do not stretch in the way that textures do, so the results vary slightly.

Select the map type from the Object Info palette; see "Applying a Texture to an Object" on page 1530.

Map Type	Description
Plane	Projects the texture onto a plane  Editing a plane map with the **Attribute Mapping** tool
Auto-Align Plane	Projects the texture perpendicularly to each face of a polygonal object; mapping is handled automatically. This type of mapping is useful for wall sculpting, and for imported 3ds (Vectorworks Design Series required) or DXF/DWG content. Do not use this map type for decals. (**Attribute Mapping** tool not supported) 
Sphere	Projects the texture to the surface of a sphere  Editing a sphere map with the **Attribute Mapping** tool

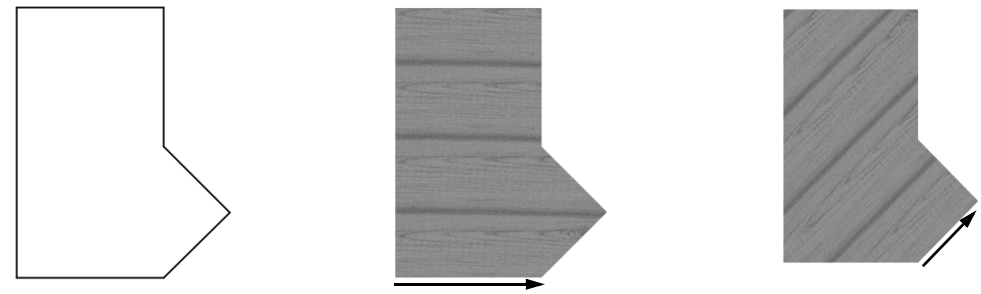
Map Type	Description
Cylinder	Projects the texture to the surface of a cylinder  Editing a cylinder map with the **Attribute Mapping** tool
Sphere and Cylinder maps have seams that are more apparent with wrapped and image-based shaders 	
Perimeter	Wraps the texture around the perimeter of the object (**Attribute Mapping** tool not supported) 
Roof	Wraps the texture across the perimeter of the roof object (**Attribute Mapping** tool not supported) 
Imported	Preserves the texture mapping from the original SketchUp or 3ds model; all other mapping controls, including the **Attribute Mapping** tool, are disabled. The map type for individual parts of an imported mesh object can be changed to other supported Vectorworks map types; it is possible to combine imported mapping and custom mapping on the same imported mesh object. The **Attribute Mapping** tool can be used for mesh parts that are set to Plane, Sphere, or Cylinder map types.

The available map types vary based on the object type being mapped.

Object Type	Map Types Available
Extrudes	Plane, Auto-Align Plane, Sphere, Cylinder, Perimeter
Meshes	Plane, Auto-Align Plane, Sphere, Cylinder, Imported
Roofs	Roof
Slabs (Floors)	Plane, Auto-Align Plane, Sphere, Cylinder
Solid Primitives (Sphere, Hemisphere, Cone)	Plane, Auto-Align Plane, Sphere, Cylinder
CSG Solids, Extrude Along Path, and Tapered Extrude	Plane, Auto-Align Plane, Sphere, Cylinder, Perimeter
Sweeps	Plane, Auto-Align Plane, Sphere, Cylinder, Perimeter
Walls	Plane, Auto-Align Plane, Sphere, Cylinder

Textures and Drawing Direction

Textures are applied to 3D polygons and walls based on the direction the object was drawn. For 3D polygons, the texture is applied parallel to the first segment.



Wall Texture Orientation

The starting point and direction the wall is drawn affects how a texture is applied. Textures are applied to a wall's left side, right side, and center (and, optionally, to its top, bottom, holes, and start and end cap). The center of a wall is always the same; the left and right sides are based on the drawing direction. When you draw a wall in a clockwise direction, which is the recommended method, the left side is the exterior side. See "Wall Direction" on page 506.

A texture applied to the center of the wall is visible at the ends of the wall.

> If the texture(s) is incorrectly applied to a wall, with the wall selected, click **Reverse Sides** on the Shape tab of the Object Info palette. This flips the direction of the wall, switching the texture(s) to the opposite side(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying a Texture to an Object
Direct Texture Mapping

Applying a Texture to an Object

To apply a texture to one or more objects:

1. Create or import a texture resource as described in "Creating Textures" on page 1505.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

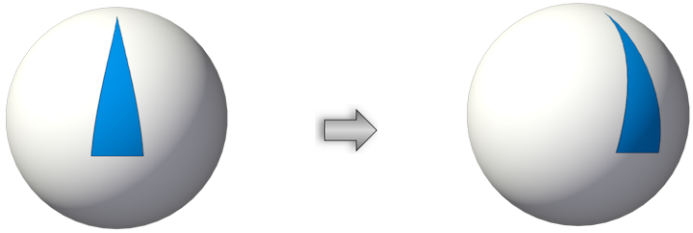
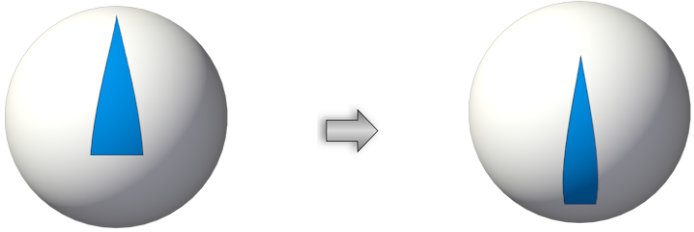
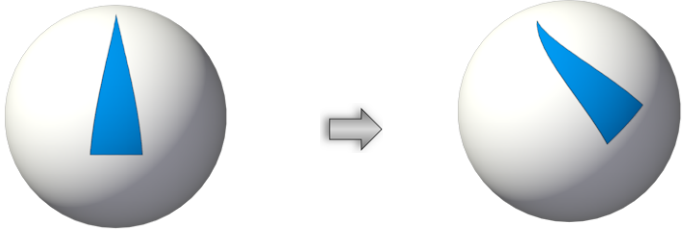
2. Select the object(s) to texture. If more than one object is selected, the texture resource applies to all objects in the selection. Additional mapping parameters are available for multiple selected objects so the texture renders seamlessly.
3. On the Attributes palette, select any **Fill Style** other than None (otherwise the texture will not render).
4. Click the Render tab on the Object Info palette.
5. If the object has parts, specify the Overall texture to apply to the parts, or select an individual part and apply a texture. The selected texture applies to the entire object if it does not contain parts.

   Select the desired **Texture** from either the default content or the current file's content (see "Resource Libraries" on page 219), and then set the texture parameters, which apply only to the selected object(s).

   The texture parameters and mapping can be edited in the Render tab of the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Part	Some objects can have different textures (including decals) applied to different parts. • For extrudes, sweeps, slabs, floors, and roof faces, apply textures to the top, bottom, and sides. • For slabs (Vectorworks Architect required), the **Mode** selection determines whether to apply textures from the slab components or from the slab. If you select By Component mode, the textures set for the slab components also determine the slab textures (see "Creating Slab Components" on page 491). If you select By Object mode, apply textures to the top, bottom, and sides. • For walls and round walls, the **Mode** selection determines whether to apply textures from the wall components or from the wall. If you select By Component mode, the textures set for the wall components also determine the wall textures (see "Creating Wall Components" on page 513). If you select By Object mode, apply textures to the left, right, top, bottom, start cap, end cap, and holes. Left and right wall texture parts can only be applied to wall holes when editing the wall hole component; see "Adding a 3D Wall Hole Component to a Symbol Definition" on page 260. • For roof objects, apply textures to the top, bottom, sides, and dormer walls, and if Vectorworks Design Series products are installed, fascia, attic, and soffit. The **Part** option is not available for objects without distinct parts. • For mesh objects imported from SketchUp or 3ds (Design Series required), the imported model determines the number of texture parts for the mesh object. A numerical list indicates how many unique textures are assigned to the imported mesh faces; the individual parts are not named. The texture or decal applied to each part displays in parentheses next to the part. To apply a texture to Overall or to a part, select it and then select a texture to apply from the texture list. Overall indicates that the same texture applies to all parts of the object. Parts that have been assigned a texture different from Overall display above the divider, and parts with the same texture as Overall display below the divider.  The Overall texture is Glass Clear, and the sides of the object are inheriting the Overall texture. The top and bottom of the object have unique textures different from the Overall texture. Overall no longer displays when all parts use distinct textures. The additional parts do not display when setting class textures (see "Applying Textures to Symbols, Walls, and Roofs" on page 1542). A wall's start cap, if set to obtain its texture by class, uses the class texture assigned to the wall center.
Revert to Overall	If a part was assigned a texture but it should inherit its texture from Overall instead, select the part and click **Revert to Overall**. The part moves back below the divider, and (from Overall) displays as its texture name. Any decals applied to the part are removed.
Add Decal	Adds a decal texture to the selected part; see "Creating Decal Textures" on page 1537
Texture list	Select the texture to apply from either the default content or the current file's content

Parameter	Description
Texture menu	Click the arrow next to the texture list to open the **Texture** menu. The texture resource or shader used to create the resource can be edited from here as described in "Editing Textures and Shaders of Selected Objects" on page 1517. Before editing, the resource can be copied to preserve the original resource parameters.
Thumbnail view	Displays a thumbnail view of the selected texture
Map Type	Select how the texture is applied to the 3D object; see "Map Types" on page 1527
Reset to Default Mapping	Removes any changes made by the **Attribute Mapping** tool or in the Object Info palette, restoring the texture to its original location, scale, and rotation.
Scale	Determines the texture size when projected onto the object. For example, a value of 2 doubles the size of the texture projection. Either enter a scale value or use the slider to change the scale. 
Offset H	Sets the start location of the texture horizontally 
Offset V	Sets the start location of the texture vertically 
Rotation	Sets the angle of texture rotation. Either enter a rotation value from 0 to 360 degrees or use the slider to change the rotation angle. 
Repeat Horizontally	Repeats the texture in a horizontal direction

Parameter	Description
Repeat Vertically	Repeats the texture in a vertical direction
Flip Horizontally	Flips the texture horizontally, along the vertical axis
Flip Vertically	Flips the texture vertically, along the horizontal axis
Radius	For sphere and cylinder map types, sets the texture radius; the default radius is the same as the 3D object radius. Increasing this value reduces the size of the texture on the object.
Follow Longest Edge	Aligns the texture's U axis with the long edge of a face, for simpler mapping when the auto-align plane map type is selected. When deselected, the texture aligns horizontally in object space.
Use World Z For Origin	Sets the texture's Z origin to be relative to the world Z origin, rather than to object space. This allows seamless texture application across multiple layers for all mapping types. Multiple objects with this parameter enabled render seamlessly along the Z axis.
Align Selected Mappings	For plane, cylinder, and sphere map types, aligns the texture mapping of multiple selected objects, giving them the appearance of a shared texture. Multiple objects with this parameter enabled use the mapping parameters of the first selected object, and the texture appears seamless across all selected objects. The shared mapping depends on the objects currently selected and can change with a different selection of objects. This parameter is not available for auto-align plane, perimeter, or roof map types.
Mesh Smoothing	Set the mesh smoothing preference for the selected object(s) to None, Use Document Preference, Custom Crease Angle, or Imported. If Custom Crease Angle is selected, specify the desired **Crease Angle**. Mesh objects imported from SketchUp or 3ds are automatically set to Imported to preserve the surface normal data from the original imported file. Imported smooth normals can be disabled and edited by selecting a different mesh smoothing option, but imported smooth normal data is deleted if the mesh is edited. This setting overrides the document preference mesh smoothing setting for the selected object(s) only. To set the document preference for mesh smoothing, see "Document Display Preferences" on page 60. This parameter is available for editing Open GL renderings in Vectorworks Design Series products even if Renderworks is not installed.
Update	Updates the drawing display

The texture can also be applied to objects from the Resource Browser. Click the texture and drag it to the desired object, or select the desired object(s), and double-click on the texture to apply. Alternatively, select the object(s), right-click (Windows) or Ctrl-click (Mac) on the texture, and select **Apply** from the context menu. However, when a texture is applied in this way, mapping options cannot be accessed directly. Click on the Render tab of the Object Info palette to edit the texture mapping.

6. The texture is applied to the selected object(s).

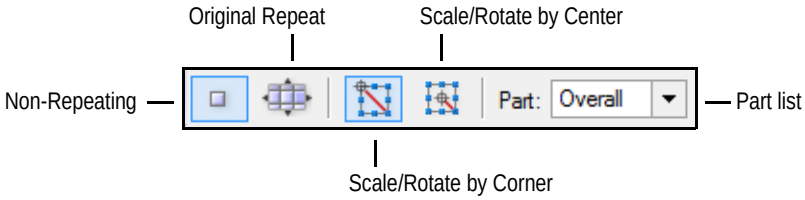
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Direct Texture Mapping
Applying and Mapping Textures
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Direct Texture Mapping

The **Attribute Mapping** tool edits texture mapping parameters directly in the drawing window. This tool applies to textures created with wrapped color shaders or mask transparency shaders. The texture mapping type must be sphere, cylinder, or plane for this tool (edit textures with other types of mapping through the Object Info palette).

This tool also modifies decal textures; see “Editing Decal Textures” on page 1540.



Mode	Description
Non-Repeating	For repeating textures, displays a single repetition of the texture for easier texture editing
Original Repeat	For repeating textures, restores the repeating pattern of the texture
Scale/Rotate by Corner	For resizing or rotating textures with the handles on the texture frame; the texture frame scales from or rotates about the handle opposite to the handle that was clicked on
Scale/Rotate by Center	For resizing or rotating textures with the handles on the texture frame; the texture frame scales from or rotates about its center
Part	For textured objects with more than one textureable part, such as walls and roofs, allows a different part of the object to be selected for texture editing; allows decals to be selected for mapping adjustments

Click here for a video tip about this topic (Internet access required).

The **Attribute Mapping** tool also edits hatch, tile, gradient, and image fills. See “Mapping Fills with the Attribute Mapping Tool” on page 1123.

To edit texture mapping directly in the drawing window:

1. Select a textured, 3D object.
2. Click the **Attribute Mapping** tool from the Basic palette or Visualization tool set. Alternatively, select the **Attribute Mapping** tool first, and then click on the object.

   If the selected object has an unsupported texture (with an auto-align plane, perimeter, or roof mapping type), an alert displays. Select another map type and click **Yes** to continue.
3. For objects with parts, which may have different textures and mapping types applied to different portions of the object, select the part to edit from the **Part** list on the Tool bar. To edit decals, see “Editing Decal Textures” on page 1540.

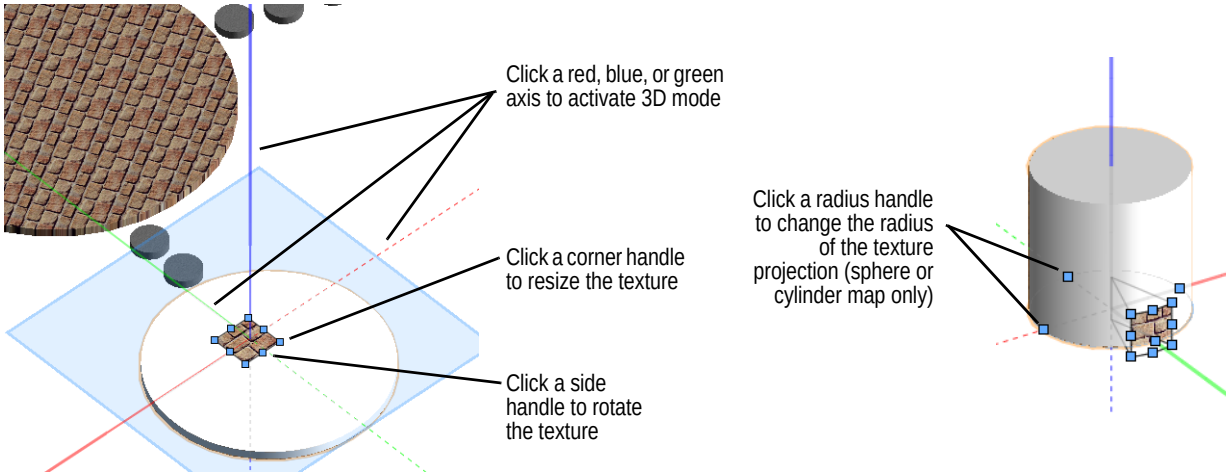
   If the selected part has an unsupported texture (with an auto-align plane, perimeter, or roof mapping type), an alert displays. Select another map type and click **Yes** to continue.
4. For easier direct mapping of a repeating texture, click **Non-Repeating** mode on the Tool bar to see a single repetition of the texture. (After you edit the texture mapping, click **Original Repeat** mode to return to the original pattern and see the effect.)

5. If applicable, select how to scale or rotate the texture from the Tool bar. To adjust the texture about a corner of the editing frame, click **Scale/Rotate by Corner** mode; to adjust the texture about the center of the editing frame, click **Scale/Rotate by Center** mode.
6. A colored texture plane with an editing frame is placed over the first repetition of the texture in 2D mapping mode. The texture origin is positioned at the origin of the axes. The Object Info palette only displays the parameters that can be edited while the **Attribute Mapping** tool is active.
   - To switch to 3D mapping mode, click one of the three texture space axes (the red, blue, and green lines).
   - To switch back to 2D mapping mode, click within the texture frame or anywhere off the textured object.

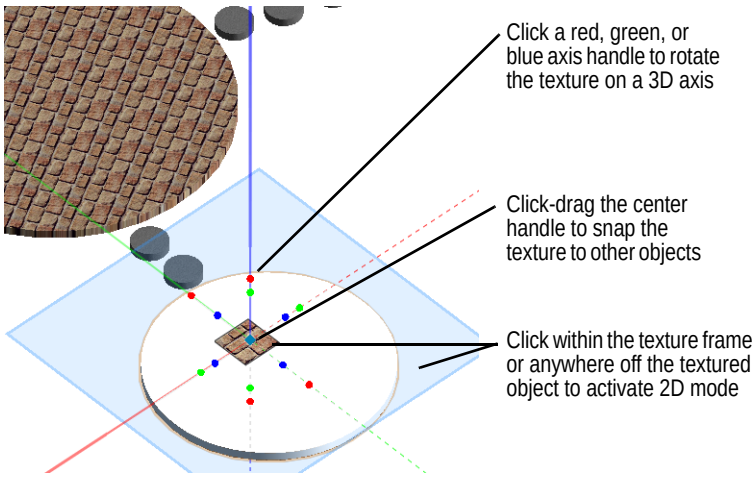
   The editing frame provides object snaps for precise texture placement. Either use the editing frame handles to adjust the mapping, or use the Data bar to enter precise values (see "Using the Data Bar" on page 125).

Action	Description
To move the texture	• 2D mapping mode: Click inside the editing frame and drag the texture to the desired location on the object surface • 3D mapping mode: Click the center diamond-shaped handle at the origin, and drag the texture to the desired location on the object surface. The texture plane moves with the image, allowing you to snap the image to a snap point on the object.
To resize the texture (2D mapping only)	Click on a corner handle of the editing frame (the resize cursor displays), and drag to set the editing object to a new scale. Click to set.
To rotate the texture	• 2D mapping mode: Click on a side handle of the editing frame (the rotate cursor displays) and drag the handle to the new rotation angle. Click to set. • 3D mapping mode: Click on one of the handles along the axis to rotate about. The rotate cursor displays, and a dashed preview indicates the rotation space. Move the cursor to rotate the texture, and click to set it.
To set the texture to a face (plane map type only)	Click the hand cursor to set the texture mapping to the surface of the indicated face. The texture plane is positioned tangent to the surface at the click point, with the origin of the texture plane at the click point.
To change the radius of the texture projection (sphere and cylinder map types, 2D mapping only)	Click on one of the four radius control handles around the edge of the circle that represents the mapping surface, and drag the handle to the new radius length. Click to set.

7. To edit the mapping of another part of a textured object, select the next part to edit from the **Part** list and repeat the process.



2D mapping mode



3D mapping mode

Applying a Texture to an Object

Creating Decal Textures

Decals allow multiple textures to be applied to a single object or part of an object, over a base texture that is already applied to the object. The decal can originate from an imported image file or a texture resource present in the file. Decals can be used to simulate signs on a textured wall, or layered textures for portraying wainscoting on a wall or patterns on fabric.

Decals must be rendered in a Renderworks mode. The Cartoon Artistic Renderworks mode cannot display decals.

To create a decal texture:

1. Select a textured, 3D object. A decal can only be applied to an object with an underlying texture; the map type of the texture determines the decal map type.

   The underlying texture must have planar, spherical, or cylindrical mapping for the decal to be edited with the **Attribute Mapping** tool. Textures with any type of mapping can be edited with the Object Info palette.

2. If the object has textureable parts, select the part where the decal will be placed from the **Part** list on the Render tab of the Object Info palette.
3. Click **Add Decal**.

   The Add Decal dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Import an Image File	Imports a new image; select the desired image file in the Import Image Document dialog box. Click **Open**.
Reuse an Existing Image	Reuses a previously imported image; select the resource that contains the image
Select a Texture	Applies the texture over the object's base texture; no further settings are required

4. If the decal consists of an imported or reused image, the Decal Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Provide a name for the decal texture
Size	Sets the real-world size for each repetition of the texture; by default, decal textures show only one repetition horizontally and vertically
Mask Options	Indicates the type of masking for the image decal
Rectangular Mask	Creates a rectangular mask around the decal; no further settings are required
Image Mask	Masks the decal image per pixel using an image mask to determine which areas of the image are transparent or opaque
Choose Image	Opens the Choose Decal Mask Image dialog box, for selecting the image mask

5. If creating a rectangular mask decal, proceed to step 9. Otherwise, if creating an image mask, click **Choose Image**.

   The Choose Decal Mask Image dialog box opens; select the image mask to use. A new image can be imported for masking, or an existing image resource from the current file can be selected (including the current image).

6. Click **OK**.

   The Create Mask dialog box opens.

Click to show/hide the parameters.

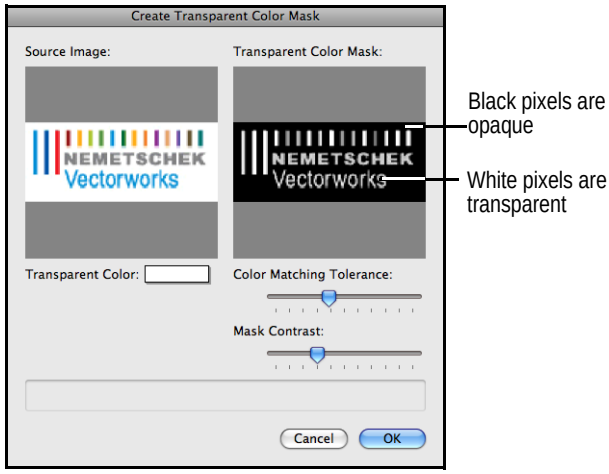
Parameter	Description
Grayscale Pixels	Creates a mask from the image's pixel brightness values; brighter pixels are more opaque
Transparent Color	Creates a mask with a selected transparent color and mask parameters

Parameter	Description
Alpha Channel	Uses the alpha channel of a source image as the mask (image must contain valid alpha channel information)

For Grayscale Pixel and Alpha Channel masks, click **OK**. Proceed to Step 9.

7. If **Transparent Color** was selected, the Create Transparent Color Mask dialog box opens. Select the mask properties and click **OK**.

   The source image must be more than eight-bit color to create a transparent color mask. Images with a monochrome background are easiest to use when creating a mask transparency.



Click to show/hide the parameters.

Parameter	Description
Source Image	Displays the imported image. Select the transparent color by clicking a color in the image; the resulting mask is displayed in the Transparent Color Mask preview. If necessary, use the mouse scroll wheel to zoom into and out of the image, or click and hold the mouse wheel button to pan.
Transparent Color	Displays the current transparent color. Instead of clicking on the source image to designate the transparent color, the color can be selected by clicking the color box.
Transparent Color Mask	Displays a preview of the mask based on the current transparent color selection and settings
Color Matching Tolerance	Adjusts the transparency tolerance; drag the slider to the right to increase the tolerance level. This allows a wider range of pixels similar to the transparent color to be considered transparent.
Mask Contrast	Adjusts the mask edge contrast; increase the contrast sharpness by dragging the slider to the right. Soften the contrast by dragging the slider to the left.

8. Click **OK** to return to the Decal Options dialog box.
9. Click **OK** to create the decal texture and apply it to the selected object.

   The decal is placed on the object; adjust its position, scale, and rotation with the **Attribute Mapping** tool or Object Info palette as described in “Editing Decal Textures” on page 1540. By default, decal textures only repeat once horizontally and vertically, with matte reflectivity and no bump or other material options. This can be modified by editing the texture from the Resource Browser.

   Click here for a video tip about this topic (Internet access required).

Editing Decal Textures

Editing Decal Textures

Similar to other textures, decal textures can be edited from the Object Info palette. Depending on map type, decal textures can also be positioned and adjusted with the **Attribute Mapping** tool. Because decals are texture resources, the texture definition can be edited from the Resource Browser.

Multiple decals can be applied to one object; the decals are applied in a stacking order, which can be changed to adjust the overall effect of the textures.

Multilayered textures and mapping parameters can be transferred between objects with the **Eyedropper** tool fill attributes (Texture). See “Transferring Attributes” on page 1095.

Decals must be rendered in a Renderworks mode.

Direct Decal Texture Mapping

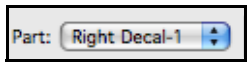
Decals can be moved, scaled, and rotated with the **Attribute Mapping** tool. Decal mapping is performed on one decal at a time, separately from the base texture mapping described in “Direct Texture Mapping” on page 1535.

The map type of the underlying texture determines the decal map type. The underlying texture must have planar, spherical, or cylindrical mapping for the decal to be edited with the **Attribute Mapping** tool.

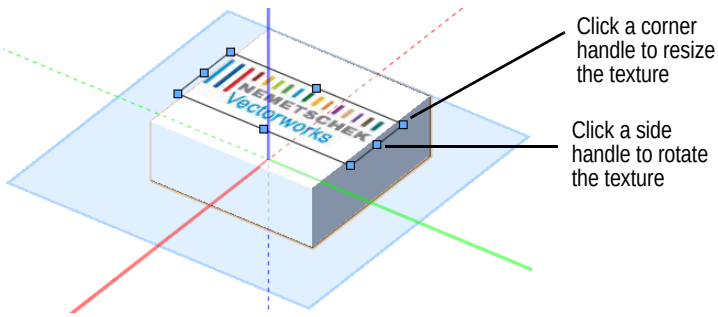
To adjust the mapping of the decal texture with the **Attribute Mapping** tool:

1. Select a textured, 3D object with one or more decals.
2. Click the **Attribute Mapping** tool from the Basic palette or Visualization tool set.

   Alternatively, select the **Attribute Mapping** tool and then click on the object with applied decal(s).
3. Select the decal to adjust from the **Part** list on the Tool bar. The Part list shows the textures mapped to the object, including the base texture.



   If the selected part has an unsupported texture underlying the decal (with an auto-align plane, perimeter, or roof mapping type), an alert displays. Select another map type and click **Yes** to continue.
4. If applicable, select how to scale or rotate the texture from the Tool bar. To adjust the texture about a corner of the editing frame, click **Scale/Rotate by Corner** mode; to adjust the texture about the center of the editing frame, click **Scale/Rotate by Center** mode.
5. An editing frame in 2D mapping mode is placed over the decal texture. Set the decal position and rotate or resize the decal as needed.



Action	Description
To move the decal	Click inside of the editing frame and drag the texture to the desired location on the base texture
To resize the decal	Click on a corner handle of the editing frame; the resize cursor displays. Drag to set the editing object to a new scale. Click to set.
To rotate the decal	Click on a side handle of the editing frame (the rotate cursor displays) and drag the handle to the new rotation angle. Click to set.

Editing the Decal Texture Parameters

When decal has been applied to an object's overall part or to its parts, its parameters can be adjusted once the decal part has been selected from the **Part** list on the Object Info palette Render tab. Fewer parameters are available for decal textures than for the base texture, because the base texture determines the mapping type for the object.

Multiple decals applied to an object add to the **Part** list and also add buttons that allow the decal stacking order to change.

Click to show/hide the parameters.

Parameter	Description
Part	Some objects can have different textures (including decals) applied to different parts; Overall indicates that the same texture applies to all different parts of the object. The texture or decal applied to each part displays in parentheses next to the part. To apply a decal to Overall or to a part, select it and then select **Add Decal**. For more information, see "Applying a Texture to an Object" on page 1530.
Remove Decal	Removes the decal from the currently selected **Part**; clicking **Revert to Overall** also removes all decals along with the base texture
Add Decal	Adds a decal to the currently selected **Part**
Move Up/Down	Decals are stacked in order as they are applied. When more than one decal has been applied to a selected **Part**, click to move the selected decal up or down in the stacking order.
Texture	Select the texture to apply from either the default content or the current file's content
Thumbnail view	Displays a thumbnail view of the selected decal
Reset Mapping	Removes any changes made by the **Attribute Mapping** tool or in the Object Info palette, restoring the decal to its original location, scale, and rotation
Scale	Determines the texture size when projected onto the object. For example, entering a scale factor of two doubles the size of the texture projection. Either enter a scale value or use the slider to change the scale.
Offset H/V	Sets the start location of the texture horizontally and vertically
Rotation	Sets the angle of texture rotation. Either enter a rotation value from 0 to 360 degrees or use the slider to change the rotation angle.
Update	Updates the drawing display

Editing the Decal Texture Definition

Decals, like other textures, are saved as resources and can be edited from the Object Info palette or the Resource Browser. Decals are masked image-based textures with transparency, but shaders can be added and shader parameters

can be changed. For example, you may wish to apply a bump shader (based on the color image shader) to letters in a sign to give them a 3D appearance. By default, the reflectivity of a decal is matte, but this can also be changed. Adjust masking parameters and add vertical and/or horizontal repetitions of the texture if the default single repetition needs to be changed.

See “Editing Textures and Shaders of Selected Objects” on page 1517 and “Editing Textures and Shaders Through the Resource Browser” on page 1519 for information on editing the decal texture and its shaders.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Decal Textures

Applying Textures to Symbols, Walls, and Roofs

While textures, and their associated surface hatches, can be applied to individual parts of many objects, as described in “Applying a Texture to an Object” on page 1530, applying textures to an object’s class or classes may be more efficient. Texture resources can be assigned to a class, which in turn can be applied to objects during creation.

Walls and roof objects have their own class texture assignment tabs for the main textureable parts of these objects. (Additional parts can be textured individually from the Object Info palette.)

> Roof faces, extrudes, sweeps, and floors contain additional textureable parts, but the Edit Class(es) dialog box does not make this distinction. Textures/surface hatches are applied by class like any other 3D object; see “Applying Object Textures by Class” on page 1542. Apply textures to the individual parts of these objects with the Object Info palette and not by class.

Textures cannot be assigned to individual symbols directly through the Object Info palette. Textures must be applied to the separate components of the symbol. This can be done using the **Edit Symbol** command or by assigning textures to the classes that make up the symbol; complex objects can contain more than one class. Texture changes affect all instances of that symbol.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Object Textures by Class
Applying Textures to Symbols
Applying Wall Textures
Applying Roof Textures
Applying a Texture to an Object

Applying Object Textures by Class

Object textures, and their associated surface hatches, can be set by class.

To apply a texture by class:

1. Select **Tools > Organization** to open the Organization dialog box.
2. Select the **Details** view option, and then select the Classes tab.
3. Select an existing class or create a new class (see “Creating Classes” on page 177).
4. With the class selected, click **Edit** to open the Edit Class(es) dialog box.
5. Select any fill **Style** other than None (otherwise the texture will not render).
6. The texture properties of the class are set in the bottom section of the dialog box. Click the Other tab.
7. Select the desired **Texture** from either the default content or the current file’s content (see “Resource Libraries” on page 219). Select **Use Textures/Surface Hatches At Creation** to apply the texture/surface hatch to objects as they are created.

   To suppress use of applied textures for the class, deselect **Texture/Surface Hatch**.

8. Click **OK** to exit the Edit Class(es) dialog box.

   The class displays on the Classes tab of the Organization dialog box.

9. If the object has not yet been drawn, click the blank area to the left of the **Class Name** to make the class active.

   If the object already exists, apply the class to it. For some object types, the class name is specified in its settings or in the Object Info palette.

10. Click **OK** to exit the Organization dialog box.
11. Create the desired object(s). Objects are created with the specified texture for that class.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Textures to Symbols, Walls, and Roofs
Applying Textures to Symbols

Applying Textures to Symbols

Textures, and their associated surface hatches, can be applied to the components that make up the symbol, or to the classes used by the symbol components.

Applying Textures to Symbol Components

To apply texture(s) to the symbol components:

1. Select the symbol to edit.
2. Select **Modify > Edit Symbol**.

   The Edit Symbol window opens; see “Editing Symbol Definitions” on page 247 for more information on the Edit Symbol window.

3. Select the first component that requires a texture.
4. On the Attributes palette, select any **Fill Style** other than None (otherwise the texture will not render).
5. Click the Render tab on the Object Info palette.
6. Select the desired **Texture** from either the default content or the current file’s content (see “Resource Libraries” on page 219), and then edit the mapping parameters as described in “Applying and Mapping Textures” on page 1527.
7. Repeat steps 3 – 6 for each component of the symbol.
8. Select **Modify > Exit Symbol** once all changes have been applied.

   The new settings are applied to all instances of the symbol.

Applying Textures to Symbol Component Classes

To apply textures to the symbol component classes:

1. Select **Tools > Organization** to open the Organization dialog box.
2. From the Classes tab, select one of the symbol classes to edit.
3. Click **Edit** to open the Edit Class(es) dialog box.
4. Select any fill **Style** other than None (otherwise the texture will not render).
5. From the Other tab, select the desired **Texture** from either the default content or the current file’s content (see “Resource Libraries” on page 219). Select **Use Textures/Surface Hatches At Creation** to apply the texture/surface hatch at creation.
6. Click **OK** to return to the Organization dialog box.
7. Repeat steps 2 – 6 until all classes have an assigned texture.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

8. Click **OK** to exit the Organization dialog box. The new settings are applied.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Wall Textures
Applying Roof Textures
Applying Textures to Symbols, Walls, and Roofs

Applying Wall Textures

Walls can be assigned a texture, and an associated surface hatch, individually with the Render tab of the Object Info palette, or, more efficiently, through an assigned class.

To apply a texture to a wall(s) by class:

1. Select **Tools > Organization** to open the Organization dialog box.
2. Select the **Details** view option, and then select the Classes tab.
3. Create a class for wall objects (see “Creating Classes” on page 177).
4. With the new class selected, click **Edit** to open the Edit Class(es) dialog box.
5. Select any fill **Style** other than None (otherwise the texture will not render).
6. The texture properties of the class are set in the bottom section of the dialog box. Click the Walls tab.
7. Select the desired **Right**, **Center,** and **Left** wall surface texture from either the default content or the current file’s content (see “Resource Libraries” on page 219). Select **Use Textures/Surface Hatches At Creation** to apply the texture/surface hatch at wall creation. See “Texture Projection and Orientation” on page 1527 to understand how textures are applied to wall faces.
8. Click **OK** to exit the Edit Class(es) dialog box.

   The new walls class displays on the Classes tab of the Organization dialog box.
9. Click the blank area to the left of the **Class Name** to make the new class active.
10. Click **OK** to exit the Organization dialog box.
11. Create the wall(s); see “Creating Walls” on page 497.

    The walls are created with the specified texture for that class. Existing wall(s) can be assigned the texture through the Object Info palette. Select the part of the wall to texture from the **Part** list and then Class Texture from the texture list on the Render tab.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Textures to Symbols, Walls, and Roofs

Applying Roof Textures

Textures, and their associated surface hatches, can be added to roof objects, dormers, and gable ends, either with the Render tab of the Object Info palette, or through an assigned class.

Use the **Roof** map type to map textures to roofs. The texture is automatically rotated for each individual roof face in the roof object.

To apply a texture to a roof object by class:

1. Select **Tools > Organization** to open the Organization dialog box.
2. Select the **Details** view option, and then select the Classes tab.
3. Create a new class for roof objects (see “Creating Classes” on page 177).

4. With the new class selected, click **Edit** to open the Edit Class(es) dialog box.
5. Select any fill **Style** other than None (otherwise the texture will not render).
6. The texture properties of the class are set in the bottom section of the dialog box. Click the Roofs tab.
7. Assign a **Top Texture**, and if being used, a **Dormer Texture** from either the default content or the current file's content (see "Resource Libraries" on page 219). Select **Use Textures/Surface Hatches At Creation** to apply the texture/surface hatch at creation.
8. Click **OK** to exit the Edit Class(es) dialog box. The class displays on the Classes tab of the Organization dialog box.
9. Click the blank area to the left of the **Class Name** to make the new class active.
10. Click **OK** to exit the Organization dialog box.
11. Create a roof with the **Create Roof** command (see "Creating Roof Objects" on page 571).

    The roof is created with the specified texture for that class. Existing roofs can be assigned the texture through the Object Info palette. Select the part of the roof to texture from the **Part** list and then Class Texture from the texture list on the Render tab.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Textures to Symbols, Walls, and Roofs

Applying Renderworks Backgrounds

Renderworks backgrounds, which include clouds, colors, images, and HDRI files, as well as a lit fog special effect, are resources saved with the file. They are applied to design layers on a per-layer basis. Backgrounds can also be applied to viewports by selecting the **RW Background** from the Object Info palette of a selected viewport. Backgrounds are also used in a variety of Renderworks styles; see "Renderworks Styles" on page 1596.

A Renderworks background can be quickly applied to the current design layer by double-clicking on the resource in the Resource Browser or by dragging it from the Resource Browser and dropping it into the current design layer.

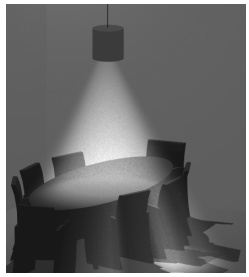
The Lit Fog weather effect requires a light source (point or spot light) in the drawing with **Lit Fog** selected in the Object Info palette.

To apply a Renderworks background to the layer properties:

1. Create the background with the desired parameters (see "Creating Layer Backgrounds" on page 1522 for information on creating Renderworks backgrounds).
2. Select **Tools > Organization** to open the Organization dialog box.
3. From the Design Layers tab, select the design layer to receive the background.
4. Click **Edit** to open the Edit Design Layers dialog box.
5. Select the **Renderworks Background** for the layer from either the default content or the current file's content (see "Resource Libraries" on page 219).

   If the Renderworks Background choices are not available, the background selection is controlled by the Renderworks style in effect. Set the background in the Renderworks style instead.
6. Click **OK** to exit the Edit Design Layers dialog box.
7. Click **OK** to exit the Organization dialog box.

   When the drawing is rendered in a Renderworks mode, the background displays. OpenGL can display One Color, Two Color, and Image backgrounds.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Lit fog effect

D Applying a Renderworks Background in Vectorworks Design Series

To apply a Renderworks background to the current layer:

1. Create the background with the desired parameters (see “Creating Layer Backgrounds” on page 1522 for information on creating Renderworks backgrounds).
2. Select **View > Set Renderworks Background** (Vectorworks Design Series required).

   The Set Renderworks background dialog box opens.
3. Select the Renderworks background for the current layer from either the default content or the current file’s content (see “Resource Libraries” on page 219).
4. Click **OK**.

   When the drawing is rendered in a Renderworks mode, the background displays. OpenGL can display One Color, Two Color, and Image backgrounds.

   When in unified view mode, the background selected here displays for all layers, overriding any background selected for individual layers in the layer properties. To retain the layer background upon exiting unified view, select **Restore Original Views when exiting Unified View mode** in the Unified View Options dialog box. See “Setting Unified View Options” on page 1152.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Layer Backgrounds

R Renderworks Shader Types

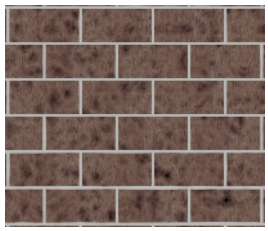
Renderworks textures are composed of four shaders: color, reflectivity, transparency, and bump. Sections on the four shader types describe each and provide a description of the parameters obtained when editing shaders.

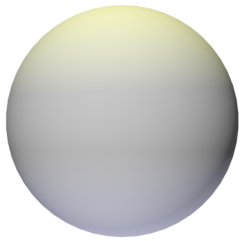
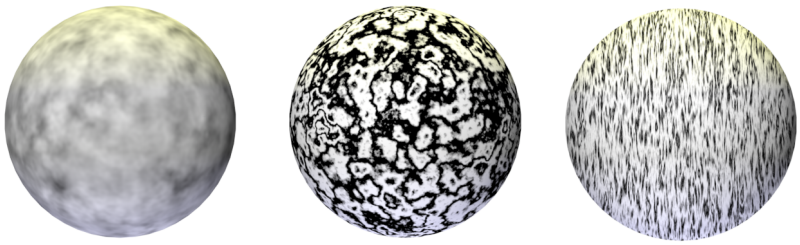
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

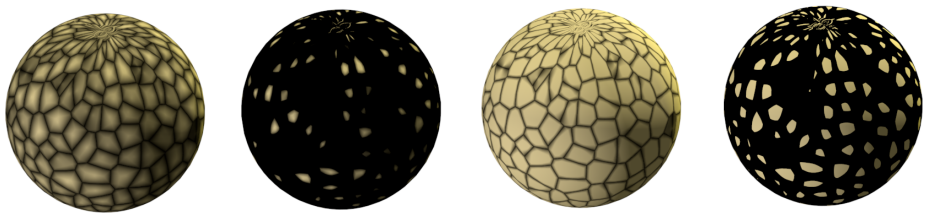
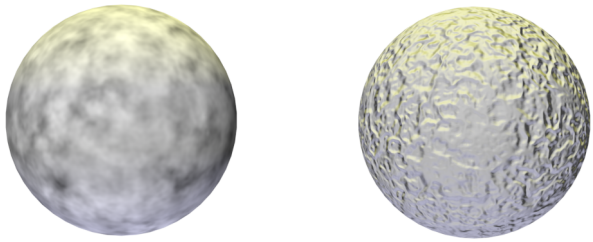
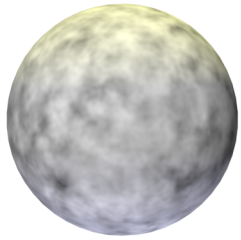
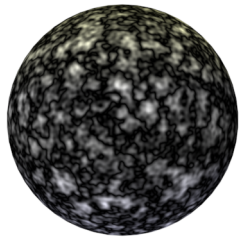
Color Shaders
Reflectivity Shaders
Transparency Shaders
Bump Shaders

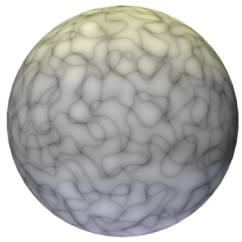
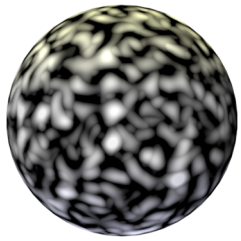
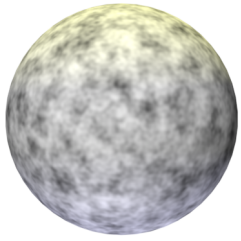
R Color Shaders

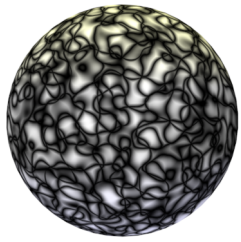
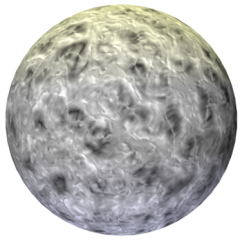
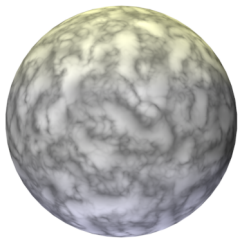
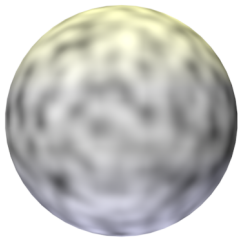
Shader	Description
Object Attribute	Selects the object’s fill color attribute to apply as a color texture
Image	Selects an imported image to apply as a color texture; the image can also be tinted with color (see “Creating Image-based Shaders” on page 1508)

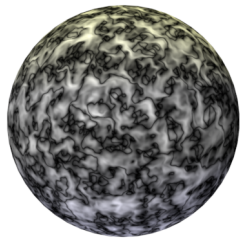
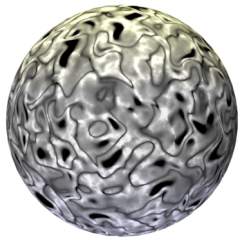
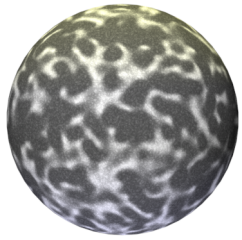
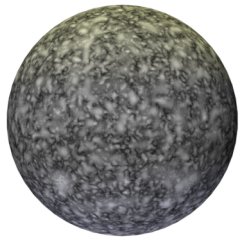
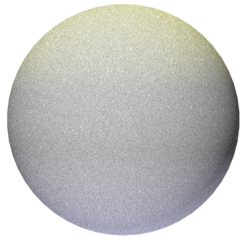
Shader	Description
Color	Specifies a color to be applied as a texture
Color	Click the color box to select the shader color
Brightness (%)	Lightens or darkens the selected color
Fresnel	Can be used to create a texture with a somewhat fuzzy appearance, like that of a pillow that appears slightly darker or lighter on its edges
Edge Color	Select the edge color by clicking the color box
Center Color	Select the main color by clicking the color box
Bricks	Creates a variegated brick pattern 
Scale (%)	Specifies the size of the bricks relative to the texture **Size**
Colors	On the Bricks, Gaps, and Alt Bricks panes, click the color boxes to select color variations within a brick/gap, or choose the same color for no variation
Choose Image	On the Bricks, Gaps, and Alt Bricks panes, click to use an image for the brick or gap texture. The chosen image is randomized to show no repetition on the textured object, so each brick/mortar joint will look realistically unique. For the completed brick surface to display correctly, the brick image should show only the flat face of a single brick or a portion of a brick, without any mortar joints. The gap image should show only mortar without any brick. Images of clay tiles and other similar materials can also be used in place of a brick.
Delete Image	Removes the selected image from the shader and uses only colors to create the variegated pattern
Image Blend (%)	When an image is used, specifies the blending between the image and the colors selected, to enhance the randomization of the resulting pattern. A low percentage value uses more color in the pattern, while a high value uses more of the image. The selected colors tint the image unless a blend value of 100% is set.
Bricks	
Contrast (%)	Sets the color contrast between the two brick colors, establishing the dot-like variation within a brick
Brick Width/ Height	Sets the brick width and height in real-world dimensions
Shift (%)	Sets the horizontal offset between rows of bricks
Half Brick Row	Adds a row of half bricks at the specified row interval
Half Brick Shift (%)	For half-brick rows, sets the amount of horizontal shift
Gaps	

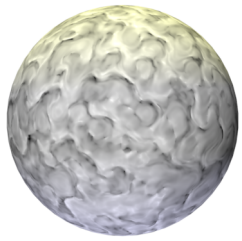
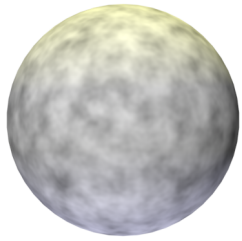
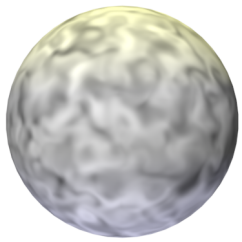
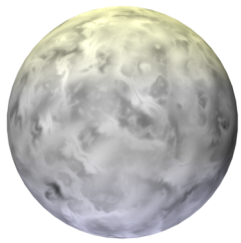
Shader	Description
Size	Sets the size of the mortar joint gap in real-world dimensions
Noise (%)	Specifies the wobbliness of the mortar joints
Alt Bricks	
Row	Adds a row of alternate brick colors at the specified row interval
Column	Adds a column of alternate brick colors at the specified column interval. **Row** must be greater than 0 to show columns.
Noise	
Pattern	Select the type of noise pattern to use. The preview helps distinguish the differences among the patterns. A generic noise pattern to select is Turbulence. Cell Voronoi is useful for creating speckled noise (set Low and High Clip around 70% each). An example of each pattern is shown using default values for a color shader on a sphere with warm lighting on the top and cool lighting on the bottom. For comparison, this sphere has no texture applied:  The pattern might look different on a plane as opposed to the sphere used for the example. In addition, the appearance of each pattern can be dramatically affected by changes to its parameters. This example shows three variations of the blistered turbulence pattern. 

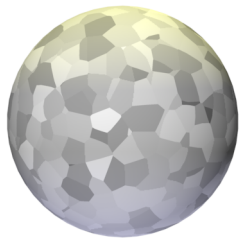
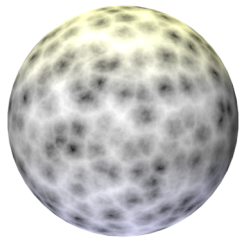
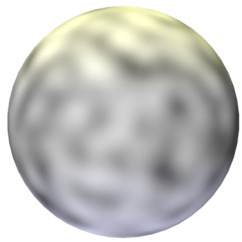
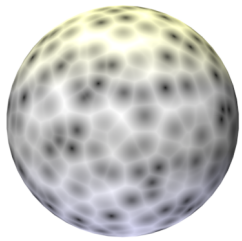
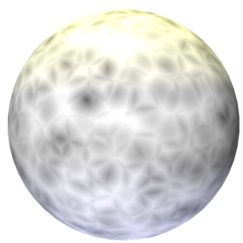
Shader	Description
Pattern (continued)	Varying only the **Low Clip** and **High Clip** percentages can completely change the appearance of the pattern.  Low Clip%: 0 High Clip%: 100 Low Clip%: 60 High Clip%: 100 Low Clip%: 0 High Clip%: 25 Low Clip%: 50 High Clip%: 50 The selected shader type also obviously affects the appearance of the pattern. This example shows the blistered turbulence pattern as a color shader and as a bump shader. 
Box	Smoothed, random cube pattern 
Blistered Turbulence	Varied distribution of light and dark areas 
Buya	Generally dark background with occasional random light areas 

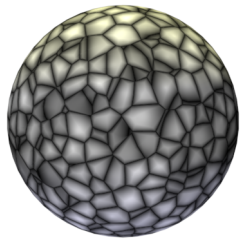
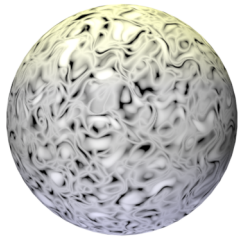
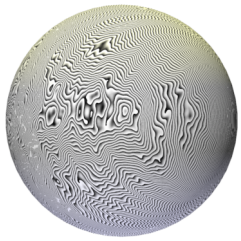
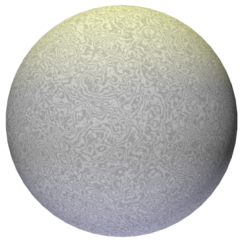
Shader	Description
Cell Noise	Tiled pattern with high contrast 
Cranal	Looping, curving pattern of lines 
Dents	Organic, twisted pattern of light and dark areas 
Displaced Turbulence	Smooth version of turbulence, with random light and dark areas 
FBM	Finer, detailed pattern of random light and dark areas 

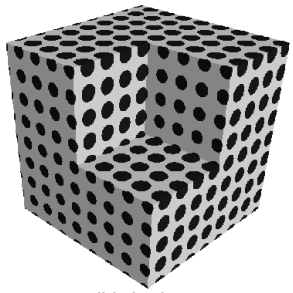
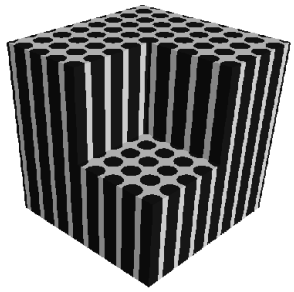
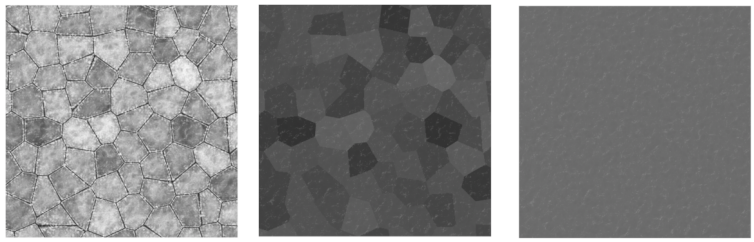
Shader	Description
Hama	Similar to the looping pattern of cranal, but more abrupt and with high contrast 
Luka	Mineral-like pattern of rough and more detailed areas 
Mod Noise	Similar to the tiled pattern of cell noise, but with less contrast 
Naki	Rough, concrete pattern 
Noise	Soft, random, and unfocused pattern of wider light areas and some dark areas 

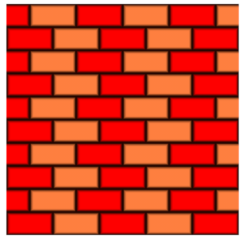
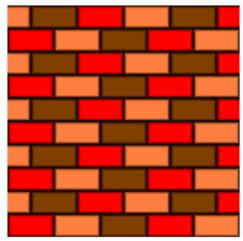
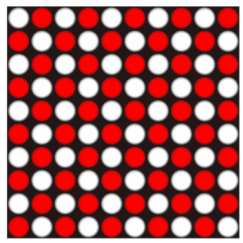
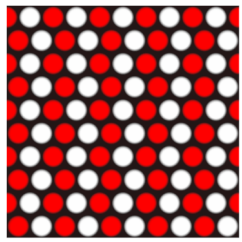
Shader	Description
Nutous	Generally dark background with smooth, flaked light areas 
Ober	A variety of rough and detailed areas, interspersed with looping, flowing lines 
Pezo	Generally dark background with smaller, patchy, lighter areas 
Poxo	Fine-grained, detailed pattern 
Random	Very fine and even detailed pattern 

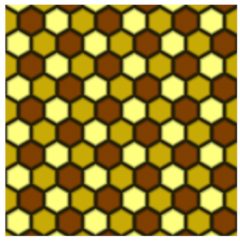
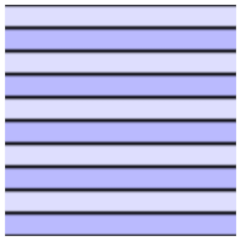
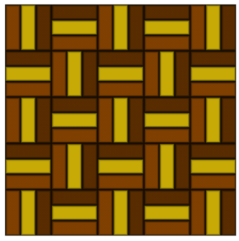
Shader	Description
Sema	Generally dark background with random, dripping/melted lines 
Stupl	Flowing, smeared pattern of light and dark areas 
Turbulence	Soft and smooth, yet detailed noise pattern 
VL Noise	Smoothed, varied, less-focused noise pattern 
Wavy Turbulence	Smooth, soft noise pattern with occasional random peaks 

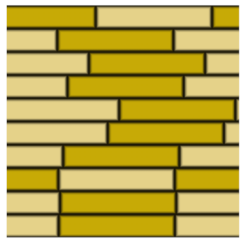
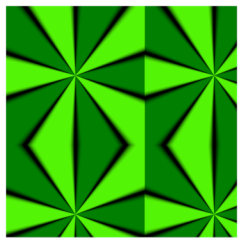
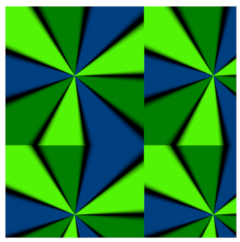
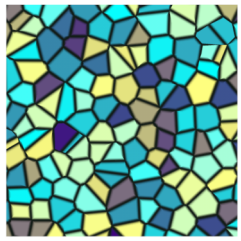
Shader	Description
Cell Voronoi	Similar to mod noise, but with irregular mosaic shapes rather than square tiles 
Displaced Voronoi	Organic voronoi pattern with smaller, inner pattern offset 
Sparse Convolution	Soft, unfocused, wide pattern of light and dark areas 
Voronoi 1	Organic, detailed voronoi pattern of small cell shapes 
Voronoi 2	Blurred voronoi pattern with offset secondary pattern 

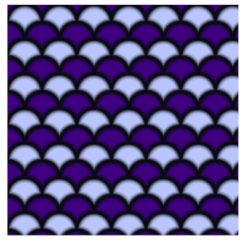
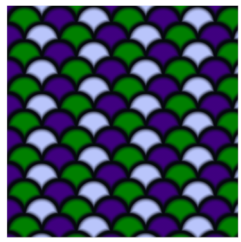
Shader	Description
Voronoi 3	Sharp organic voronoi pattern of irregular cell shapes, high contrast with dark outlines 
Zada	Twisting, organic pattern with sharp looping areas and flat areas 
Wood	Detailed wood-grain pattern 
Marble	Marbled, veined pattern with high detail 
Colors	Click the color boxes to select color variations within the noise pattern
Scale	
Global (%)	Sets the size of the pattern as a percentage of the texture **Size**
Relative (%) U, V, W	Allows the pattern to be stretched in the U, V, and/or W direction; can be used to simulate the appearance of scratches
Options	

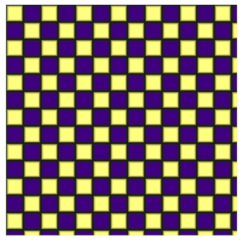
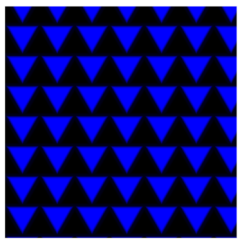
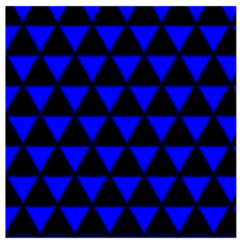
Shader	Description
Dimensionality	Select 2D Wrapped or 3D Solid. Wrapped shaders are 2D patterns projected onto a 3D object surface. Solid shaders are 3D patterns applied to a 3D object surface; when the shape changes, the object and pattern are still displayed correctly.  Solid shader  Wrapped shader
Detail	Sets the level of detail or resolution for many of the patterns
Cycles	Creates a repeating pattern of banded noise texture; set the number of repetition cycles to use
Low/High Clip (%)	The low and high clip determine the abruptness of the transition between noise colors and are often set as a combination to achieve a desired effect. Set the Low clip higher or the high clip lower for a sharp transition to the other color; set the clips evenly to make the transition between noise colors more even.
Pavement	Creates the appearance of cracked pavement; with no gaps, simulates variegated pavement, and with the same two colors, simulates solid pavement 
Scale (%)	Sets the size of the pavement as a percentage of the texture **Size**
Stones	
Colors	Click the color boxes to select color variations within the pavement, or choose the same color for no variation
Coarse Marbling (%)	Increases or decreases the coarse marbling appearance of the stones
Fine Marbling (%)	Increases or decreases the fine marbling appearance of the stones
Gaps	
Colors	Click the color boxes to select color variations within the pavement joints
Width (%)	Sets the width of the pavement joint gap as a percentage of the texture **Size**
Softness (%)	Sets the bevel amount between pavement stones and pavement joints
Grain (%)	Sets the amount of color variation within the pavement gaps
Smudges	

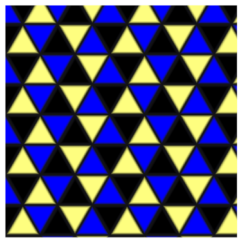
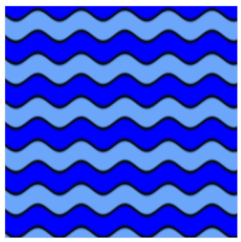
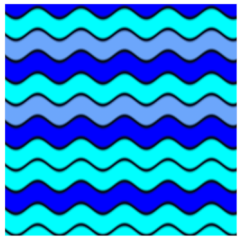
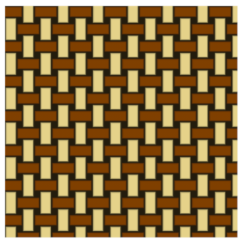
Shader	Description
Colors	Click the color boxes to select color smudging variation within the pavement gaps
Amount (%)	Sets the amount of smudging for the pavement gap smudges
Size (%)	Increases or decreases the amount that the smudging extends from the gap centers
Tiles	
Pattern	Select the type of tile pattern. The preview helps distinguish the differences among the patterns. Some of the patterns use two of the tile colors, while others make use of all three.
Brick 1	Alternating brick pattern with two colors 
Brick 2	Alternating brick pattern with three colors 
Circles 1	Alternating polka dot pattern with two colors 
Circles 2	Offset polka dot pattern with two colors 

Shader	Description
Circles 3	Offset polka dot pattern with three colors 
Hexagons	Alternating hexagon pattern with three colors 
Lines 1	Alternating pattern of lines with two colors 
Lines 2	Line pattern with three colors 
Parquet	Parquet pattern with three colors 

Shader	Description
Planks	Alternating rectangular pattern, similar to wood planks, with two colors 
Radial Lines 1	Starburst pattern with two colors 
Radial Lines 2	Starburst pattern with three colors 
Random	Irregular cellular pattern with blends of three colors for a stained glass effect 
Rings 1	Alternating circular pattern with two colors 

Shader	Description
Rings 2	Alternating circular pattern with three colors 
Sawtooth 1	Alternating zig-zag pattern with two colors 
Sawtooth 2	Zig-zag pattern with three colors 
Scales 1	Alternating scale pattern with two colors 
Scales 2	Alternating scale pattern with three colors 

Shader	Description
Spiral 1	Alternating spiral pattern with two colors 
Spiral 2	Alternating spiral pattern with three colors 
Squares	Alternating checkerboard pattern with two colors 
Triangles 1	Alternating triangle pattern with two colors 
Triangles 2	Offset triangle pattern with two colors 

Shader	Description
Triangles 3	Offset triangle pattern with three colors 
Waves 1	Alternating wavy pattern with two colors 
Waves 2	Wavy pattern with three colors 
Weave	Basketweave pattern with two colors 
Colors	Click on each color box to select the color for the grout, and up to three colors for the tiles
Randomize Colors	Uses randomly-selected colors for the pattern, mixing the selected colors in no particular order
Dimensions	
Grout Width (%)	Sets the width of the grout joint as a percentage of the texture
Bevel Width (%)	Sets the bevel amount between tiles and grout
Horizontal Orientation	When selected, tiles are oriented horizontally; deselect for vertical tiles
Scale	
Global (%)	Sets the size of the tile pattern as a percentage of the texture **Size**

Shader	Description
Relative (%) U, V	Sets the relative scale of the tile pattern in the U (width) or V (height), allowing the tiles to be stretched

Reflectivity Shaders
Transparency Shaders
Bump Shaders
Creating and Mapping Textures

Reflectivity Shaders

Shader	Description
Image	Specifies an imported image to be used as the reflectivity shader (white is more reflective; black is less reflective); see “Creating Image-based Shaders” on page 1508
Backlit	This shader is useful for lamp shades, light bulbs, and curtains that are lit from behind. Deselect **Cast Shadows** in the Edit Texture dialog box when using the backlit shader.
Color	Specifies a color to be applied as a backlit color; click the color box to select the color
Brightness (%)	Controls the backlit brightness
Reflection (%)	Sets the amount of reflection to use, or set to 0 for no reflection
Blurriness (%)	Sets the amount of blurriness for the reflection, or set to 0 for no blur
Glass	
Edge Color	Click the color box to set the color applied to the glass at an angle, which is seen at the edge of the glass
Center Color	Click the color box to set the color at the center (main) part of the glass
Blurriness (%)	Sets the amount of blurriness for the reflection, or set to 0 for no blur
Glow	The glow shader offers an alternative to creating line or area lights, and can create “neon” signs
Brightness (%)	Sets the amount of glow
Emit Light	Allows the texture to become a light source, when Indirect Lighting is enabled (**View > Lighting > Set Lighting Options**)
Add Matte Reflectivity	Creates a partially glowing light that is also lit by other light sources
Reflection (%)	Sets the amount of reflection to use, or set to 0 for no reflection
Blurriness (%)	Sets the amount of blurriness for the reflection, or set to 0 for no blur
Metallic	Creates metallic effects (also try colored mirror shaders for polished metal effects). Metallic effects require the presence of light objects in the drawing. HDRI background lighting will not produce metallic effects.
Color	Click the color box to select the metallic color

Shader	Description
Pattern	Select the type of metallic pattern • Metallic: provides a metallic paint effect (such as for car bodies) • Brushed: provides a brushed metal appearance • Turned: creates a machine-turned round pattern
Reflection (%)	Sets the amount of reflection to use, or set to 0 for no reflection
Blurriness (%)	Sets the amount of blurriness for the reflection, or set to 0 for no blur
Mirror	
Color	Click the color box to select the mirror color; use grayscale for a regular mirror effect, and select colors for mirrored metallic effects
Reflection (%)	Sets the amount of reflection to use, or set to 0 for no reflection
Blurriness (%)	Sets the amount of blurriness for the reflection; using a non-grayscale color and some blurriness creates an effective metallic look
Plastic	
Color	Click the color box to select the plastic color
Brightness (%)	Sets how bright the shader appears
Roughness (%)	Sets the width of shiny areas on the plastic; increase the percentage for wider shiny areas
Reflection (%)	Sets the amount of reflection to use, or set to 0 for no reflection
Blurriness (%)	Sets the amount of blurriness for the reflection, or set to 0 for no blur
Bricks	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Noise	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Pavement	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Tiles	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546

Color Shaders
Transparency Shaders
Bump Shaders
Creating and Mapping Textures

Transparency Shaders

Shader	Description
Image Mask	Selects an imported image to apply as a transparency mask (see “Creating Image-based Shaders” on page 1508)

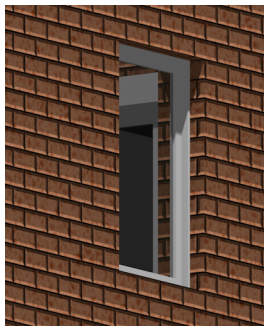
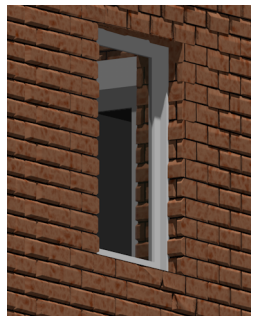
Shader	Description
Image	Selects an imported image to apply as a transparency texture (see “Creating Image-based Shaders” on page 1508), for colored transparency and projected colored lights like gobo projections in the Vectorworks Spotlight software
Color	Specifies a color to be applied as a transparency
Color	Click the color box to select the transparency color
Brightness (%)	Lightens or darkens the selected color
Glass	When using glass transparency, set the accompanying color shader to a dark color (or even black) for best results
Transmission (%)	Sets how much light passes through the glass; set to a high value when the glass is very clear
Index of Refraction	As light moves through a medium, the index measures the change in the direction of the light’s rays. An index of 1.0 indicates no change; a typical value for water and ice is 1.3, and for glass use 1.5 – 1.6. To save time, use a value of 1.0x (just above 1) when the glass is thin.
Color	Click the color box to select the color that appears in a very thin object; for glass, this is usually set to white
Blurriness (%)	Blurs the transparency by the specified amount, or set to 0 for no blur. Blurred glass appears frosted (and takes longer to render).
Absorption Color	Click the color box to select the color that a large, thick object assumes, when a ray of light has traveled the **Absorption Distance**  Different absorption colors (green, blue, and black, with the Color set to white)
Absorption Distance	Defines the distance rays of light have to travel before **Absorption Color** replaces **Color**. The lower the value, the more intense the **Absorption Color** is. This applies to thick or thin glass.  Different absorption distances set, from low to high
Plain	Provides plain, uniform transparency of a specified amount

Shader	Description
Opacity (%)	Sets how opaque the shader is; set to a lower value for more transparency and to a higher value for more opacity
Rectangular Mask	Uses the texture as a rectangular mask to apply as a transparent texture (usually combined with another shader to create a specific masking effect). Useful for decals.
Horizontal/Vertical Repetitions	Indicates how to display the mask: a single instance, infinite repetitions, or a set number of repetitions in each direction
Bricks	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Noise	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Pavement	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Tiles	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546

Color Shaders
Reflectivity Shaders
Bump Shaders
Creating and Mapping Textures

Bump Shaders

Shader	Description
Image	Specifies an imported image to be used as the source of the bump map displacement (see “Creating Image-based Shaders” on page 1508)
Bricks	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Noise	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Pavement	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546
Tiles	These parameters are the same for all shaders; for a description, see “Color Shaders” on page 1546

Shader	Description
Displacement Mapping	For realistic bump textures, displacement mapping creates texture and bumpy details with a rendering technique that appears embossed, projecting the geometry outward from the surface.  Bump shader without displacement mapping  Bump shader with displacement mapping This mapping applies only to Final Quality and Custom Renderworks, when **Displacement Mapping** is enabled in the render options. Rendering can be significantly slower with displacement mapping. If an image bump does not provide the desired results, try a noise bump shader.
Height	Specify a non-zero height to enable displacement mapping; large height values may result in longer render times
Detail	Sets the level of detail for displacement mapping; requirements and results vary depending on the texture and the surface's face size. Textures without too much bump detail and a large face size, such as boards or stones, render with less detail and can be set with a lower level of detail; fine, faceted textures like grass or leaves may require a high level of detail, which also requires more rendering time. Conversely, very large surfaces, like a ground plane, may need higher levels of detail to see the displacement.
Self-Shadowing	Adds shadows to the displaced geometry, increasing realism as well as rendering time

Color Shaders
Reflectivity Shaders
Transparency Shaders
Creating and Mapping Textures

Rendering the Drawing

Rendering Overview

Rendering gives a solid appearance to 3D objects. Solid objects can cast shadows and occlude (hide) objects that are behind them.

The Vectorworks Fundamentals product provides a quick rendering solution using OpenGL (see “Rendering with Vectorworks” on page 1587). OpenGL performs lighting calculations for fast, interactive rendering and medium-quality render previews, but it cannot create reflections. For final-quality output, use the Renderworks product; it provides additional light sources, and it can cast shadows, create reflections, add indirect lighting effects, and produce much better rendering results through intensive lighting calculations. Render settings can be saved as a Renderworks style for re-use, and styles can be shared among files. The Renderworks product includes the Visualization palette for easily accessing all lights and cameras in the file. Another benefit of using the Renderworks product is that an image can be rendered in any mode from a specified portion of the drawing.

Optimizing Rendering Performance

Rendering is often a memory-intensive and time-consuming process. Modify your model and change rendering options in these ways to increase rendering performance and speed:

- Use the highest-level geometry possible to model your objects. It is much better to model an object as an extrude, sweep, Boolean solid, or a NURBS surface than as a mesh or a set of 3D polygons. For extrude and sweep profiles, use polylines, and do not use a sweep segment angle smaller than five degrees. Examine imported geometry for excessive vertices and polygons. The slowest way to model for rendering is to create a detailed object as a set of small, individual 3D polygons.
- Eliminate 3D details that are not significant to the rendered image or the indirect lighting. For example, if you model a staircase's individual bolts and threaded screws, each of these will be contributing unnecessarily to the rendering time. Use classes to hide geometry that is not going to be visible in the rendered view.
- Test rendering settings on sheet layer viewports with lower sheet layer resolution values. Sheet layers can render in the background while you continue working. Sheet layers are set to 72 dpi by default, but for testing, set the dpi value even lower (such as 40). Zoom out of a drawing to reduce the rendered area, or use the **Render Bitmap** tool (Renderworks required) with a low dpi setting.
- Use sheet layer viewports and render bitmaps for final renders, since they can render in the background (Renderworks render modes only).
- Consider turning off ambient lighting (**Ambient Info** off in the Lighting Options or set Ambient Light dialog box) to evaluate the contribution of individual light sources more clearly.
- Change rendering options temporarily to test a rendered look. For OpenGL options, disable **Use Anti-Aliasing** (Mac only). In the Custom Renderworks options (Renderworks required), set the **Quality** levels to low and turn **Anti-Aliasing**, and even **Shadows,** off. Render with Fast Renderworks to evaluate before switching to Final Quality or Custom Renderworks.

- Shadows add to rendering time. Shadows can also be turned off for light objects that do not cast obvious shadows. In Renderworks, individual textures can be set not to cast or receive shadows (**Shadows** options in the Edit Texture dialog box). A clear glass texture, for example, has very slight shadows and does not cast shadows on other objects. In addition, enable soft shadows only for light sources that create prominent shadows in an image (for example, a directional light that represents the sun).
- When using indirect lighting (Renderworks required), use the fewest number of bounces needed (as specified in the Lighting Options dialog box) and adjust the quality in the Final Quality or Custom Renderworks settings. Individual textures can have their own indirect lighting setting overrides to reduce the rendering time required; disable indirect lighting for these textures.
- Area and line lights (Renderworks required) contribute significantly to rendering time. Use these light sources sparingly, and adjust the **Quality** of each in the Object Info palette. Only enable **Soft Shadows** for these lights if their shadows are prominent in the image.
- Reflective, blurry reflective, blurry transparent, and glow shaders add to rendering time (Renderworks required). Use these effects sparingly. Blurriness is faster at less than 15% and should not be set to more than 40% (for wide, shiny highlights, consider using a plastic or metallic reflectivity shader rather than a blurry reflection). Glowing objects need to be sampled more accurately and therefore, they take more time. Use the glow shader for a few, large objects; use the backlit shader for several small fixtures with simple light sources.
- In Renderworks, glass adds to render times because it has reflective, refractive, and transparent shadow components.
  - Very clear glass can be set to not cast shadows in the Edit Texture dialog box to save render time. Indirect lighting can be disabled for glass textures in the Edit Texture dialog box, as well, to save time when calculating indirect lighting.
  - For thin panes, like windows, the refractive qualities of glass are not significant, so the index of refraction parameter for the glass can be set to a low value (just above 1).
  - Often, buildings use glass with a metallic/mirrored energy efficient coating; in that case, a combination of mirror reflectivity and plain transparency can produce a glass effect that renders faster.
  - Glass absorption color can be used for very accurate close-ups of items like glassware, but this effect takes longer to render.
  - Blurry (frosted) glass, in particular, can be slow; use blurry glass in a strictly controlled way. Lower blur values render faster.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Light
Advanced Renderworks Lighting
Renderworks Features for OpenGL
Preparing to Render
Background Rendering
Rendering with Vectorworks
Rendering with Renderworks
Rendering a Selected Area
Batch Rendering
Managing Lights and Cameras with the Visualization Palette
Exporting a Rendered Drawing

Adding Light

Once one or more light sources have been added to the drawing, it can be rendered to mimic the effect of light on the drawing surfaces.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Default lighting is automatically added to a drawing for basic visibility of rendered objects. The default lighting is fixed to the "camera" so that it always lights an object appropriately. However, rendering a complex scene usually requires the addition of light sources; the addition of a visible light automatically hides the default lighting.

Light added to a Vectorworks drawing can be ambient or diffused. Ambient light affects all surfaces equally. Diffused lighting affects surfaces differently depending on the angle of the light source and position of the surfaces.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Ambient Light and Sunlight
Adding Light Sources

Ambient Light and Sunlight

Setting Lighting Options

The **Set Lighting Options** command specifies the ambient light brightness and default color for the current layer, or, when **Unified View** is selected, the view. By default, the ambient light parameters are set to **On**, with a **Color** of white, and a **Brightness** of 35%. When the Renderworks product is installed, this command also sets the white balance for any light sources with a color temperature, and specifies environment background lighting parameters when a Renderworks HDRI background resource has been added to the drawing.

The ambient light of a selected viewport can be edited by clicking **Lighting Options** from the Object Info palette (see "Creating Sheet Layer Viewports" on page 1616).

To set ambient light:

1. Switch to the layer where the desired ambient light settings are to be set.
2. Select **View > Lighting > Set Lighting Options**.

   The Lighting Options (Renderworks required) or Set Ambient Light dialog box opens. Set the ambient light for the layer. The white balance setting can be adjusted for light sources in the layer with a color temperature specified (Renderworks required). For a chart of common color temperature ranges, see "Correlated Color Temperature" on page 1852.

   If the lighting options are controlled by a Renderworks style that is currently in effect, the Edit Renderworks Style dialog box opens instead; see "Renderworks Styles" on page 1596.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Indirect Lighting (Renderworks required)	Indirect lighting models the transfer of light energy among drawing objects and materials, as the light bounces around and affects the surroundings. The more bounces specified, the more realistic the rendering, but also the more time required to render. Select the level of indirect lighting to use for rendering: • **None**: Direct lighting only, for quick rendering • **Exterior, 1 Bounce**: One bounce of lighting is often enough to enhance exterior views • **Normal, 2 Bounces:** Two bounces of indirect lighting is appropriate for exterior scenes and well-lit interiors • **Interior, 4 Bounces**: Using four bounces of indirect lighting is suited for darkened or partially-lit interior scenes, or very detailed and complete indirect lighting Individual textures can have their own indirect lighting setting overrides to enhance a scene while reducing the rendering time required. When rendering with Fast Renderworks, indirect lighting is rendered at a "Low" quality; Final Quality Renderworks uses a "High" quality setting for indirect lighting, and Custom Renderworks allows the quality setting to be selected.
Ambient Info	
On/Off	Activates or deactivates the ambient light settings for the layer
Brightness	Specifies the ambient light brightness; enter a percentage or drag the slider bar
Color	Specifies a default color associated with the ambient light; click the color box to select the color
Emitter Options (Renderworks required)	
Emitter Brightness (%)	Adjusts the brightness of all light objects with **Use Emitter** selected
White Color Temperature	Select a color temperature that will be balanced to appear white. Lower temperatures, which might otherwise have an orange cast, or higher temperatures, which might otherwise appear with a blue tint, are adjusted to appear white. Select **Custom** to specify a temperature to be white balanced.
Custom (K)	If a Custom **Color Temperature** is specified, enter the temperature in Kelvin
Environment Lighting (HDRI) (Renderworks required)	When an HDRI layer background has been selected for the layer or viewport, specifies how to control its lighting contribution to the rendering (see "Creating HDRI Backgrounds" on page 1524)
From Current Background	Uses the image environment background set for the layer as both a background and a light source
From Selected Background	Lights the model with the colors from the HDRI background selected here
None	Uses the image environment background set for the layer or viewport as a background only. The background does not contribute to lighting.

3. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Light
Adding Sunlight
Shadow Analysis

Adding Sunlight

Sunlight in Vectorworks Fundamentals is created by projecting parallel rays from a directional light.

The Vectorworks Design Series products have expanded capabilities with the **Heliodon** tool; see “Solar Studies” on page 1489.

To set the sun position:

1. Select **View > Lighting > Set Sun Position.**

   The Set Sun Position dialog box opens. Enter the sun position information.

Click to show/hide the parameters.

Parameter	Description
Site	Specifies the site’s latitude and longitude
Time Zone Meridian	Specifies the site’s time zone; one hour of difference is equal to 15º of longitude
Page North	Normally, the drawing is oriented to page north (true north, not magnetic north), and true north matches the top of the page, with no compensation required for proper sun position. If the drawing was not created with this orientation, specify an angular offset in degrees from page north. The North indicator should match true north in the drawing.
Date	Specifies the time of year
Time	Specifies the time of day
Daylight Savings	Select if Daylight Saving time is in effect
Results	Based on the settings made, displays the **Azimuth** (South at zero degrees, without Page North rotation), **Azimuth N. Relative** (North at zero degrees, without Page North rotation), **Elevation** (degrees above the horizon), and **True Solar Time** (takes into account daylight savings time, site longitude and time zone, and the earth’s orbit and tilt)

2. Click **OK** and verify the results. If a directional light was selected before choosing the **Set Sun Position** command, the command modifies the angles of light to the new azimuth and elevation. If no light was selected, the tool inserts a new directional light at the specified sun position.

   A shadow analysis can be performed for a site by inserting several light sources with identical parameters except for time of day. The Renderworks product must be installed to conduct a shadow analysis.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Light
Shadow Analysis
Applying Colors

R Shadow Analysis

With the Renderworks product, a shadow analysis can help determine the areas of the drawing that will receive less sunlight or remain in a shadow during a particular time of the year. By inserting several directional light sources, each representing a different time of day, an overlapping range of shadows is created which indicates the amount of time a particular area is in the shade over the course of the day.

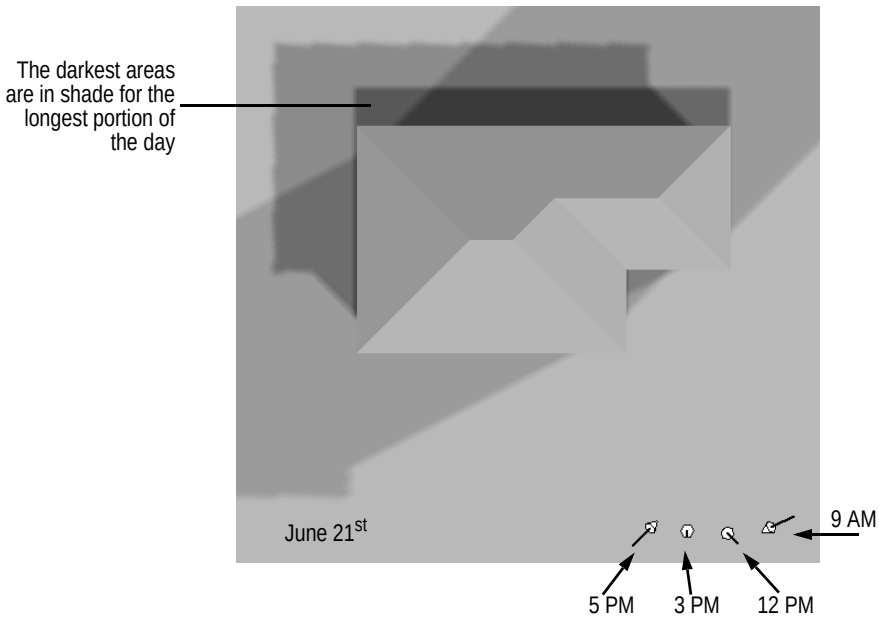
To perform a shadow analysis:

1. Add the objects to the drawing that will cast a shadow on the site (a house, for example).
2. Orient the drawing to page north. If the drawing was not created with this orientation, this can be compensated through the Set Sun Position dialog box described in the next step.
3. Insert each directional light source by selecting **View > Lighting > Set Sun Position**. Inserting one light source provides shadow information for that time of day only; several light sources set to different times of day yield an actual shadow analysis with overlapping shadows.

   The Set Sun Position dialog box opens. See “Adding Sunlight” on page 1573

   For each light source inserted with this command, specify the same parameters except for time of day. At a minimum, a light source for morning, noon, afternoon, and evening should be inserted.
4. Verify the parameters of each directional light source in the Object Info palette. Ensure that **Cast Shadows** is selected, and decrease the **Brightness** so that the combination of lights is not too harsh.
5. Render the drawing with the **Fast Renderworks**, **Custom Renderworks**, or **Final Quality Renderworks** option. Use the **Render Bitmap** tool to render a portion of the image for presentations.

   See “Rendering a Selected Area” on page 1608 for more information on the **Render Bitmap** tool.



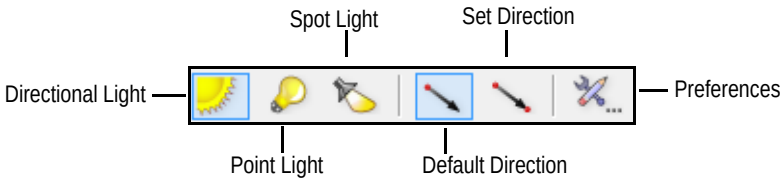
Top view – Shadow analysis

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Adding Light
Adding Sunlight
Solar Studies

Adding Light Sources

The **Light** tool places light sources in the drawing. Select the type of light and specify the light preferences from the Tool bar.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



When the Renderworks product is installed, additional parameters and light source types are available. These additions are described in this section and in “Advanced Renderworks Lighting” on page 1582.

Mode	Description
Directional Light	Projects light with parallel rays, like the sun
Point Light	Radiates light in all directions, like a bare light bulb
Spot Light	Projects light in a specific direction, aimed at a specific object, like a flashlight or conventional spotlight
Default Direction	For directional lights, click to specify the light position
Set Direction	For directional lights, click to set the light direction, and then click to specify the light position
Preferences	Sets the preferred light parameters

Adding a Light Source
Light Source Properties

Adding a Light Source

Adding a visible light source to a drawing hides the default lighting scheme that is automatically present for basic rendering purposes.

To add a light source:

1. Click the **Light** tool from the Visualization tool set.

   From the Tool bar, select the type of light source to insert (directional light, point light, or spot light). If inserting a directional light, select the light placement method (Default Direction or Set Direction).

2. Click the **Preferences** Tool bar button to specify the light source preferences for this session. Depending on the light source selected, different parameters are available. In addition, certain parameters are only available when the Renderworks product is installed.

   The Light Preferences dialog box opens. Set the parameters.

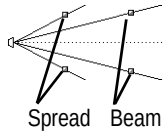
Click to show/hide the parameters.

Parameter	Description
On/Off	Shows or hides the light produced by the light source
Color	Specifies a color associated with the light source; click the color box to select the color. This parameter is not available if **Use Emitter** is selected and a **Color Temperature** is specified (Renderworks required).
Cast Shadows (Renderworks required)	Select to create shadows
Soft Shadows (Renderworks required)	Creates more realistic shadows by decreasing hard edges; shadows appear softer the farther they travel, as in reality. Rendering is slower when this option is enabled.
Brightness	Specifies the light source brightness; enter a percentage or drag the slider bar. A value over 100% can be entered. This parameter is not available if **Use Emitter** is selected (Renderworks required).
Use Emitter (Renderworks required)	For accuracy, specifies the light's actual brightness and color temperature; leave deselected to use the light as a simple light source
Get Brightness From (Renderworks required)	Specifies the luminous quantity of a light
User Input (Renderworks required)	Specifies the brightness as an accurate number in Lux, Lumens, Footcandles, or Candelas; the units vary depending on the light source
Distribution File (Renderworks required)	Does not apply to directional, point, or spot lights (see "Advanced Renderworks Lighting" on page 1582)
Color Temperature (Renderworks required)	Specifies the light color temperature in Kelvin. This refers to an ideal black body emitter, glowing "red hot" or "white hot." A lower temperature generates an orange color; the hotter the temperature, the closer to white the color of the light is. Specifying this parameter is optional. If not specified, the default temperature is 0, meaning that the final emission color for the light is entirely controlled by the selection in **Color**. When the temperature is specified, **Color** cannot be changed. The final emission color is set by the **Color Temperature**. Color temperature settings can be white-balanced on a per-layer basis; see "Setting Lighting Options" on page 1571.
Directional, Spot, or Point Light Specs	Click to set additional specifications for the light source

3. Click **Directional Light Specs**, **Spot Light Specs**, or **Point Light Specs**, for the selected light source type, to specify additional parameters.

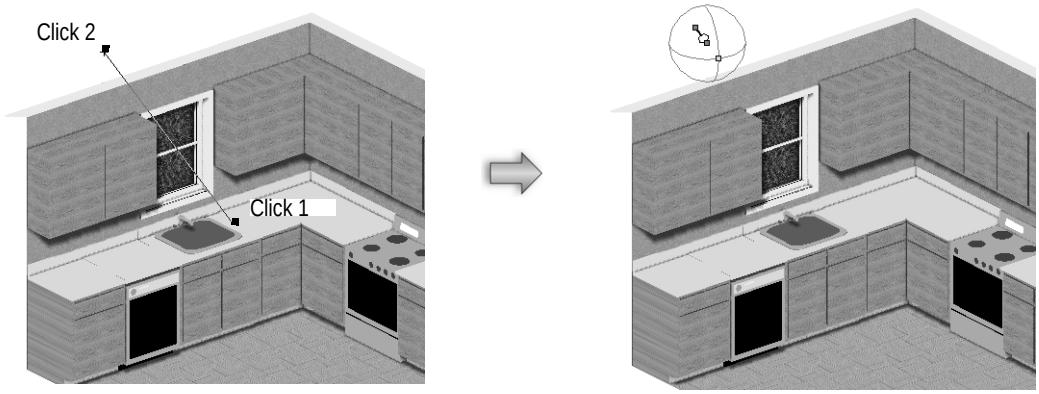
Click to show/hide the parameters.

Parameter	Description
Directional Light	

Parameter	Description
Direction	Specifies the light's direction by either specifying the light angle or vector
Angle	Sets the light's angle by azimuth and elevation. The azimuth angle is set based on an angle of 0 degrees at the negative Y axis, and is positive in a counter-clockwise direction; the elevation angle is the angle above (positive) or below (negative) the horizon.
Vector	Indicates the direction of the light by specifying the coordinates of its X, Y, and Z vectors
Point Light	
Dist Falloff	Select the distance falloff function (rate of intensity change while moving along the beam away from the light source): • None: Brightness does not change • Smooth: Brightness decreases according to a linear falloff • Realistic: Light is brightest near the light source, and falls away according to the square of the distance, as in reality. This falloff selection is the most natural in appearance.
Spot Lights	
Spread/Beam diagram	Drag the handles on the diagram to set the spot light **Spread** and **Beam** angles, or enter the values in the fields below the diagram 
Spread	Specifies the spread angle of the spot light (light cone's maximum angle)
Beam	Specifies the beam angle of the spot light (cone of light that does not change intensity up to the spread angle)
Dist Falloff	Select the distance falloff function (rate of intensity change while moving along the beam away from the light source): • None: Brightness does not change • Smooth: Brightness decreases according to a linear falloff • Realistic: Light is brightest near the light source, and falls away according to the square of the distance, as in reality. This falloff selection is the most natural in appearance.

4. Click **OK** to return to the Light Preferences dialog box. Click **OK** to return to the drawing.
5. Click to place a light object with the parameters specified in the Light Preferences dialog box.

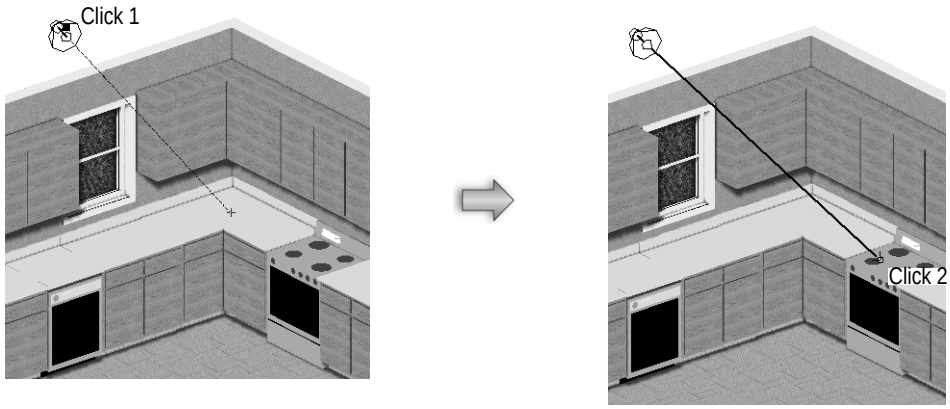
   If placing a directional light, click to specify the light position in Default Direction mode. In Set Direction mode, click once to specify the light target or direction, and then click a second time to specify the light position.



Set Direction mode of directional light placement depicted

If placing a spot light, click to place the light, and then drag to specify the light direction and target. The spot light can be aimed at any object. Click again to set the spot light. The target Z height can be adjusted precisely with the **Look To Height** parameter of the Object Info palette.

The spot light target handle and projection line only display when the spot light is selected. Use the **Selection** tool to move the light. The target handle aims the spot light and can be adjusted with the **Selection** tool once the spot light has been created. Use the **Reshape** tool to move the target handle constrained about an axis selected in the Tool bar.



Adding Light Sources
Light Source Properties
Managing Lights and Cameras with the Visualization Palette

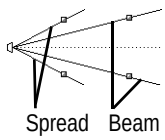
Light Source Properties

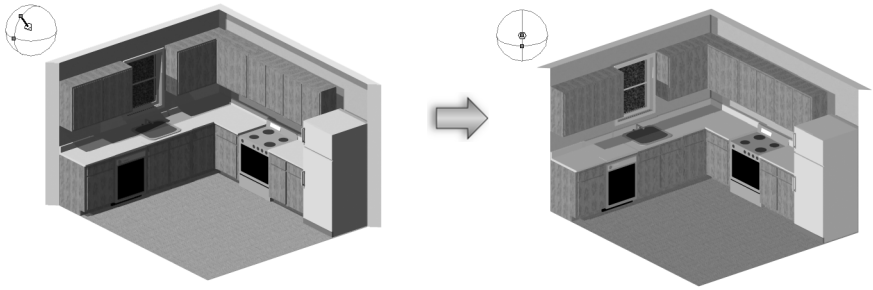
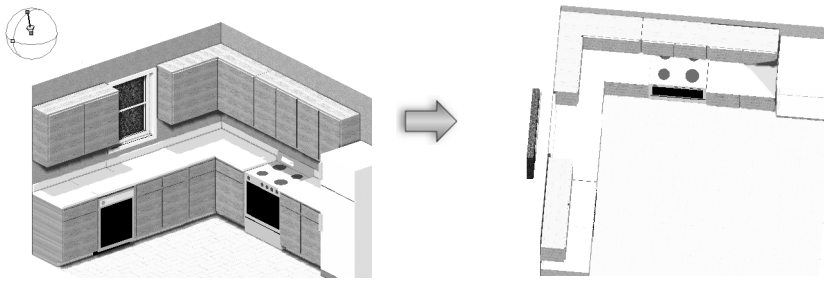
Light source parameters are displayed and can be modified in the Object Info palette. The parameters available depend on the type of light source.

Some of the parameters are only available when the Renderworks product is installed; see “Advanced Renderworks Lighting” on page 1582 for information on the Renderworks light types.

Click to show/hide the parameters.

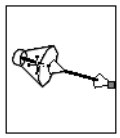
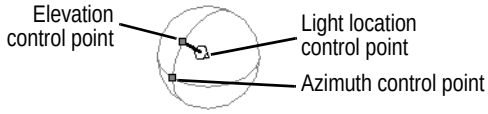
Parameter	Description
Kind	Indicates the type of light source; change to a different type by selecting a different kind from the displayed list A custom, area, or line light requires that the Renderworks product be installed.
On/Off	Shows or hides the light produced by the light source. This setting can also be accessed from the light context menu: right-click (Windows) or Ctrl-click (Mac) on the light, and select **Turn On** or **Turn Off**.
Auto Update	Automatically renders the drawing when rendering-related parameters change
Update	When **Auto Update** is deselected, click to render the drawing when rendering-related parameters change
Cast Shadows (Renderworks required)	Select to create shadows
Soft Shadows (Renderworks required)	Creates more realistic shadows by decreasing hard edges; rendering is slower when this option is enabled
Lit Fog (Renderworks required)	Creates a special volumetric lighting effect for point lights and spot lights when used in combination with the Lit Fog Renderworks background weather effect (see “Creating a Background” on page 1522)
Use Emitter (Renderworks required)	For accuracy, specifies the light’s actual brightness and color temperature; leave deselected to use the light as a simple light source. Required for a custom, area, or line light. The emitter brightness of all light objects in a scene with **Use Emitter** selected can be controlled in Lighting Options; see “Setting Lighting Options” on page 1571.
Brightness	Specifies the light source brightness; enter a percentage or drag the slider bar. A value over 100% can be entered. This parameter is not available for custom, area, and line lights, or if **Use Emitter** is selected (Renderworks required).
Brightness From (Renderworks required)	Specifies the location of the light intensity data for a custom light (either user input or distribution file); the distribution file can be specified by clicking **Load Distribution**
Brightness Value (Renderworks required)	Specifies the brightness as an accurate number
Brightness Unit (Renderworks required)	Select Lux, Lumens, Footcandles, or Candelas for the unit of Brightness; the available units depend on the light type
Dimmer (Renderworks required)	Dims the light source brightness (intensity); enter a percentage or drag the slider bar. Only the brightness of the light source is affected; the color temperature is not changed.
Light Rotation Angle (Renderworks required)	Specifies the rotation angle of a custom light source around an axis connecting the light location to the light target; this angle defines the plane for the intensity distribution curve, and is displayed in red on the custom light object

Parameter	Description
Use Distribution File (Renderworks required)	Specifies whether to use the attached emission profile data file for the custom light
Distribution File (Renderworks required)	For a custom light, displays the distribution file name when a valid distribution file has been selected, or "None" if a valid distribution file has not been designated (click **Load Distribution** to specify a file)
Load Distribution (Renderworks required)	For a custom light, loads light emission profile data from a standard file. The brightness value is obtained using the integral of the raw emission data provided with the file. The file must be a text file with industry standard intensity distribution data in .ies format.
Color Temperature (Renderworks required)	Specifies the light color temperature in Kelvin. This refers to an ideal black body emitter, glowing "red hot" or "white hot." A lower temperature generates an orange color; the hotter the temperature, the closer to white the color of the light is (see "Correlated Color Temperature" on page 1852 for typical light source color temperature ranges). Specifying this parameter is optional. If not specified, the default temperature is 0, meaning that the final emission color for the light is entirely controlled by the selection in **Color**. When the temperature is specified, **Color** cannot be changed. The final emission color is set by the **Color Temperature**. Color temperature settings can be white-balanced on a per-layer basis; see "Setting Lighting Options" on page 1571.
Color	Specifies a color associated with the light source; click the color box to select the color. This parameter is not available if **Use Emitter** is selected and a **Color Temperature** is specified (Renderworks required).
Dist Fall	Indicates the distance falloff function for a point, spot, custom, area, or line light (specifies the rate of intensity change while moving along the beam away from the light source)
Quality (Renderworks required)	For area and line lights, specifies the sampling quality of the light; select From Render Mode to use the settings specified for **Soft Shadows** quality in the rendering options
Render Geometry (Renderworks required)	For area and line lights, renders the light object geometry
Spread/Beam diagram	Drag the handles on the diagram to set the spot light **Spread** and **Beam** angles, or enter the values in the fields below the diagram 
Spread	Specifies the spread angle of the spot light (light cone's maximum angle)
Beam	Specifies the beam angle of the spot light (cone of light that does not change intensity up to the spread angle)
X/Y/Z or X'/Y'/Z'	Specifies the location of the light source in relation to the active layer plane or working plane
Look To Height	For spot lights, sets the Z height above the active layer plane of the spot light target handle

Parameter	Description
Azimuth	Indicates the azimuth for a directional light (South at zero degrees); enter new values or drag the slider bar to set
Elevation	Indicates the elevation (degrees above the horizon) for a directional light; enter new values or drag the slider bar to set
Pan/Tilt 	Click the top button to display the spot light or custom light beam pan and tilt angles; enter new values or drag the slider bar to set
±X/±Y/±Z 	Click the bottom button to display the light beam direction in terms of its X, Y, and Z components
Set Light to View	Sets the orientation of the directional, spot, or custom light to that of the current view. This setting can also be accessed from the light context menu: right-click (Windows) or Ctrl-click (Mac) on the light, and select **Set Light to View**. 
Set View to Light	Sets the orientation of the current view to that of the directional, spot, or custom light. This setting can also be accessed from the light context menu: right-click (Windows) or Ctrl-click (Mac) on the light, and select **Set View to Light**. 

A symbol that contains a light object can be copied to reproduce identical lights. Change the light intensity for each symbol in the Object Info palette.

Once a light has been placed on the drawing, the **Selection** tool can be used to change its location, and, depending on the light type, beam parameters.

Light Type	Beam Parameter	Description
Point, Spot, Directional	Light location	Click on the light with the move cursor and drag it to its new location  The spot light target handle is not locked to its target. The target handle location may need re-adjusting after moving the spot light.
Spot, Directional	Orientation and view	Select the light; in the Object Info palette, click **Set Light to View** to set the light orientation to the current view orientation, or click **Set View to Light** to set the orientation of the view to that of the light
Directional	Azimuth and elevation direction	Click on the azimuth or elevation control points to change the directional light parameters. Move the handles with the mouse, or enter the azimuth and elevation values in the Data bar. 
Spot	Target direction	Click on the handle with the **Selection** tool to change the spot light target direction; drag to the new target and click to set 
Directional	Beam direction	Click and drag with the hand cursor to change the beam direction 

Managing Lights and Cameras with the Visualization Palette

Advanced Renderworks Lighting

Default lighting is added by the Vectorworks program for the basic visibility of rendered objects. For a more realistic rendering, one or more light sources can be added to the drawing. The addition of a light source automatically hides the default lighting scheme, so that the scene is not overly bright.

The brightness and color of objects in shadow are affected by the layer's ambient light setting.

The Vectorworks Fundamentals product provides three light types: directional, point, and spot. With the Renderworks product, additional parameters are provided for these light types, and additional light source types are included.

Renderworks product rendering modes must be used for the Renderworks lighting options.

For information on the lighting types and parameters in the Vectorworks Fundamentals product, see “Adding Light” on page 1570.

Light Type	Description	Product
Directional	Projects light with parallel rays	Vectorworks and Renderworks
Point	Radiates light in all directions	Vectorworks and Renderworks
Spot	Projects light in a specific direction	Vectorworks and Renderworks
Line	Emits light from a line	Renderworks only
Area	Emits light from the surfaces of objects	Renderworks only
Custom	Emits light based on a defined complex spatial distribution	Renderworks only

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting an Area or Linear Light
Inserting a Custom Light
Light Source Properties
Rendering with Renderworks

Inserting an Area or Linear Light

An area or linear light can be created from an existing object or line. Unlike the other light source types, which are emitted from a specific, concentrated source, area and linear lights emanate from an extended object. This is useful for creating diffused light sources with softer lighting, such as fluorescent lights, neon lights, windows, and skylights.

An area light cannot be textured. Use a glow reflectivity shader to create textured objects that emit light only when indirect lighting is enabled in the Lighting Options dialog box.

The addition of multiple diffused light sources can add significantly to rendering time. For efficiency, replace a small area or line light with a point, spot, or directional light. In addition, since area lights can be slow to render, it may be better to use an Image Environment (HDRI) or physical sky background to supply the background (sky) lighting.

To create an area or linear light from an existing object or objects:

1. If creating an area light, select the object to convert to a light source; area light geometry should enclose a surface (a line or open polyline cannot be used, for example). Select a 2D line, closed 2D surface, or NURBS curve if creating a linear light.

   Select more than one object to create several line or area lights at the same time.

   Curved surfaces used for an area light require more rendering time than planar surfaces.

2. Select **Modify > Convert > Convert to Area Light** or **Modify > Convert > Convert to Line Light**.

   The Light Preferences dialog box opens; indicate the area or line light parameters. Specify light intensity by entering a **Get Brightness From** value.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
On/Off	Shows or hides the light produced by the light source
Color	Specifies a color associated with the light source; click the color box to select the color. This parameter is not available if a **Color Temperature** is specified.
Cast Shadows	Creates shadows
Soft Shadows	Creates more realistic shadows by decreasing hard edges; rendering is slower when this option is enabled
Dimmer (%)	Dims the light source brightness (intensity); enter a percentage or drag the slider bar. Only the brightness of the light source is affected; the color temperature is not changed.
Use Emitter	An area or line light's actual brightness and color temperature must be specified, so this setting is not optional
Get Brightness From	Specifies the luminous quantity of the area or line light
User Input	Manually specifies the luminous quantity of an area light as an accurate number; the units of a line light are always Lumens
Distribution File	For custom lights only; does not apply to area or line lights
Color Temperature	Specifies the light color temperature in Kelvin. This refers to an ideal black body emitter, glowing "red hot" or "white hot." A lower temperature generates an orange color; the hotter the temperature, the closer to white the color of the light is (see "Correlated Color Temperature" on page 1852 for typical light source color temperature ranges). Specifying this parameter is optional. If not specified, the default temperature is 0, meaning that the final emission color for the light is entirely controlled by the selection in **Color**. When the temperature is specified, **Color** cannot be changed. The final emission color is set by the **Color Temperature**. Color temperature settings can be white-balanced on a per-layer basis; see "Setting Lighting Options" on page 1571.
Area or Line Light Specs	Sets specific area or line light options

3. Click **Area Light Specs** or **Line Light Specs** to set additional parameters.

   The Area Light Data or Line Light Data dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Dist Falloff	Select the distance falloff function (rate of intensity change while moving along the beam away from the light source)
Quality	For area and line lights, specifies the sampling quality of the light; select From Render Mode to use the settings specified for **Soft Shadows** quality in the rendering options
Render Geometry	Renders the light object geometry; deselect to hide the original geometry when rendering

4. Click **OK** to return to the Light Preferences dialog box.

5. Click **OK**. The object or line is converted into a light. If more than one object was selected for conversion, the converted objects are grouped. The area light emits light in both surface directions.

   The light must be rendered with a Renderworks product rendering mode. When an area light is on and rendered, it has constant reflectivity and does not receive shadows; when it is off, it displays as a normal object.

   If a line light with **Cast Shadows** enabled is not casting light as expected, the center of the line light's bounds may be inside another object, which prevents shadow calculations. If this occurs, either deselect **Cast Shadows**, use several line lights, or split the line light geometry into segments with their bounding centers located outside of another object.

Editing an Area or Linear Light

The light parameters can be edited in the Object Info palette.

The area or linear light original geometry can be edited.

To edit the original light geometry:

1. Select the area or line light and then select **Modify > Edit Light**.

   The Edit Light window opens, and the original 3D object or a NURBS representation of a 2D object is displayed.
2. The object geometry can be edited with the **Reshape** tool.
3. Click **Exit Light** at the top right of the drawing window to return to the drawing.

R Inserting a Custom Light

A custom light source's emission distribution can be defined by a standard intensity distribution profile for accurate physical lighting.

To create a custom light:

1. Select the **Light** tool from the Visualization tool set, and then select **Custom Light** from the Tool bar.
2. Specify the custom light parameters by selecting **Preferences** from the Tool bar.

   The Custom Light Data dialog box opens. Click **Load Distribution** and specify the location of the custom light distribution file, and then specify any additional custom light parameters.

Click to show/hide the parameters.

Parameter	Description
Light Rotation Angle	Specifies the rotation angle of the light source around an axis connecting the light location to the light target; this angle defines the reference plane for the intensity distribution curve
Distribution File	Displays the distribution file name when a valid distribution file has been selected, or "None" if a valid distribution file has not been designated (click **Load Distribution** to specify a file)
Load Distribution	Loads light emission profile data from a standard file. The brightness value is obtained using the integral of the raw emission data provided with the file. The file must be a text file with industry standard intensity distribution data in .ies format.
Dist Falloff	Select the distance falloff function (rate of intensity change while moving along the beam away from the light source)
Direction	Specifies the light's direction by either specifying the light angle or vector

Parameter	Description
Angle	Sets the light's angle by pan and tilt. The pan angle is based on an angle of 0 degrees at the positive Y axis, and is positive in a counter-clockwise direction; the tilt angle is equal to 0 at the horizontal plane, positive when pointing below the plane, and negative when pointing above the plane.
Vector	Indicates the direction of the light by specifying the coordinates of its X, Y, and Z vectors

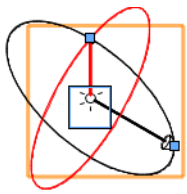
3. Click **OK**.

   The Light Preferences - Custom Light dialog box opens. Specify additional custom light parameters. Emitter parameters are specified by the distribution file, and cannot be changed.

Click to show/hide the parameters.

Parameter	Description
On/Off	Shows or hides the light produced by the light source
Color	Specifies a color associated with the light source; click the color box to select the color. This parameter is not available if a **Color Temperature** is specified.
Cast Shadows	Creates shadows
Dimmer	Dims the light source brightness (intensity); enter a percentage or drag the slider bar. Only the brightness of the light source is affected; the color temperature is not changed.
Use Emitter	A custom light's actual brightness and color temperature must be specified, so this setting is not optional
Get Brightness From	Specifies the luminous quantity of the area or line light as a manually entered value or from a loaded distribution file
User Input	Manually specifies the brightness in Lux or Lumens
Distribution File	Sets intensity distribution data with a standard file specified in **Custom Light Specs**. The brightness value is obtained using the integral of the raw emission data provided with the file.
Color Temperature	Specifies the light color temperature in Kelvin. This refers to an ideal black body emitter, glowing "red hot" or "white hot." A lower temperature generates an orange color; the hotter the temperature, the closer to white the color of the light is (see "Correlated Color Temperature" on page 1852 for typical light source color temperature ranges). Specifying this parameter is optional. If not specified, the default temperature is 0, meaning that the final emission color for the light is entirely controlled by the selection in **Color**. When the temperature is specified, **Color** cannot be changed. The final emission color is set by the **Color Temperature**. Color temperature settings can be white-balanced on a per-layer basis; see "Setting Lighting Options" on page 1571.
Custom Light Specs	Specifies additional custom light options and loads a distribution file

4. Click **OK** to return to the drawing.
5. Click in the drawing to insert the custom light.



The custom light object is represented by a pair of perpendicular arrow-head vectors and two perpendicular circles. The black vector points to the target location; its axis line (the light axis) connects the light source location to the target. The red vector starts at the light source location, pointing to a reference point on the "equator" of the polar intensity distribution. Also known as the "zero angle line," it represents the origin for measuring the intensity on the light curve.

The two vectors form the black circle, and the red circle is perpendicular to it. The black circle represents the original plane where the light curves are located. The red circle constrains the movement of the zero angle line.

After a custom light has been placed, the light parameters can be edited in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Light Source Properties

Preparing to Render

Once the drawing is complete or near completion, any textures have been applied, and the desired lighting has been added, select one of the standard views (see "Using Standard Views" on page 1141), and use the **Flyover**, **Walkthrough**, **Translate View**, or **Rotate View** tools to orient the drawing. For presentation purposes, it is often easier to create sheet layer viewports and render the sheet layers, or create render bitmaps; both sheet layer viewports and render bitmaps can render in the background (Renderworks required) while drawing processes can continue.

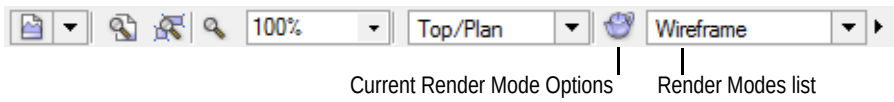
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Rendering with Vectorworks
Rendering with Renderworks
Background Rendering
Optimizing Rendering Performance
Rendering a Selected Area
Batch Rendering

Rendering with Vectorworks

The render modes in the Vectorworks program translate the drawing in various ways to create an image with additional realistic details. For example, a Hidden Line rendering is similar to the non-rendered (Wireframe) image, but it hides the portion of the objects that would not normally be visible. An OpenGL rendering also has hidden lines, and colors and shading; it also shows how the light sources interact with the object surfaces, and it can show any textures that were applied.

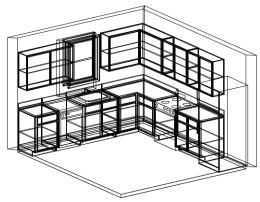
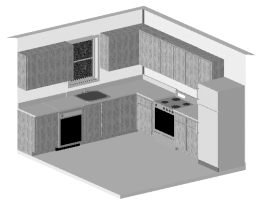
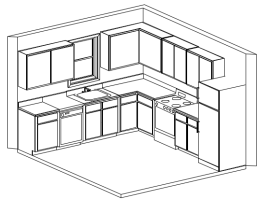
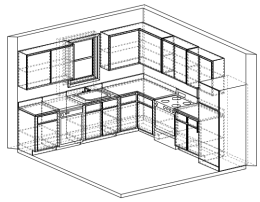
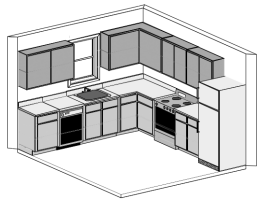
The View bar displays the **Render Modes** for quick access to the rendering commands. The View bar options display the Render Mode - Short (displaying a shortened version) or the Render Mode - Long (displaying the current render mode options and the current render mode). The **Current Render Mode Options** button opens the settings dialog box for the current render mode, if applicable.

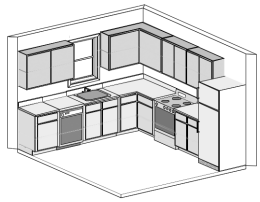
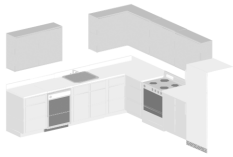
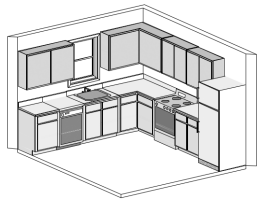


To select a render mode:

Select **View > Rendering**, and then select the desired render mode.

To cancel a render process before it is complete, press Esc.

Rendering Command	Description
Wireframe	A visual representation of the model is created using lines to represent object edges; see “Wireframe Options” on page 1589 to adjust settings for planar objects 
OpenGL	Creates a good-quality, detailed rendering, with colors, shading, and textures (optional); see “OpenGL” on page 1590 for details 
Hidden Line	Hides the edge lines of objects that are behind other objects, which gives a solid appearance; with the Renderworks product installed, optionally displays surface hatches that give the appearance of textures (see “Hidden Line Rendering with Surface Hatches” on page 1606) 
Dashed Hidden Line	Edge lines of objects that are behind other objects display as dashed lines 
Unshaded Polygon	Displays objects as solids, and displays attributes such as colors 

Rendering Command	Description
Shaded Polygon	An Unshaded Polygon rendering with shading added 
Shaded Polygon - No Lines	A Shaded Polygon rendering with no edge lines 
Final Shaded Polygon	A Hidden Line rendering on top of a Shaded Polygon - No Lines rendering; edges and curved surfaces are cleaner than those in a Shaded Polygon rendering 

A Vectorworks preference sets the preferred 3D render mode, applied whenever the drawing is switched from Top/Plan into a 3D view; see “3D Preferences” on page 54.

If you have a Vectorworks Service Select subscription, you can use the Vectorworks Nomad app to view and present your 3D drawings rendered in OpenGL on your iOS mobile device. Click here for more information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Vectorworks Cloud Services
Line Render Options
Rendering with Renderworks

Wireframe Options

Planar objects with solid fills are drawn in stacking order. In a 3D wireframe view, objects may overlap, obscuring objects lower in the stacking order. The wireframe options control whether fills should be visible for planar objects in 3D views, and if they are visible, sets the fill opacity of filled planar objects.

To set wireframe options:

1. Select **View > Rendering > Wireframe Options**.

   The Wireframe Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Show fills in planar objects in 3D views	Displays fills for 2D planar objects in 3D views
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Fill opacity	Sets the opacity of filled 2D planar objects for situations where objects overlap
Ignore perspective	When selected, draws quick orthographic representations of PDFs, images, and image fills (including tile definitions which contain images or image fills); deselect to draw more accurate perspective-projected representations of PDFs, images, and image fills, which may take more time
Ignore perspective in text	When selected, draws quick orthographic representations of text in 3D views; deselect to draw accurate perspective-projected text, which may take more time to represent

2. Click **OK** to set the wireframe options.

OpenGL

Use the OpenGL render mode to create good-quality rendering previews that are fast and interactive. (For final-quality output, use the Renderworks product; see “Renderworks Rendering Modes” on page 1593.)

2D planar objects and graphics render with full fill attributes.

The Vectorworks program has a default lighting scheme, so that a basic rendering does not require an added light source. However, the addition of a light source is usually necessary for a more realistic rendering (see “Adding Light Sources” on page 1574). The addition of a light source automatically hides the default lighting scheme, so that the scene is not too bright.

OpenGL renders up to eight lights in a drawing; additional ambient light, directional, point, or spot lights have no effect. Custom, area, and linear lights can be added (Renderworks required).

To get the best performance from OpenGL, use a video card that supports hardware-accelerated OpenGL. There are significant compatibility issues with some cards; see www.vectorworks.net for a list of video cards that are compatible with the Vectorworks program. If a model is too complex to render the design layer successfully, an alert message displays, informing you that the render mode has been switched to wireframe.

The power-saving video settings of some laptops may cause out-of-memory issues or other display issues that can easily be solved by turning off the power-saving video features.

Click here for a video tip about this topic (Internet access required).

OpenGL Options

Use the OpenGL options to control the level of detail in rendered images, which in turn affects the render speed (less detail renders faster). These settings apply only to the current drawing; they remain in effect in the current drawing until the settings are changed. The current OpenGL settings are saved when you create a template (see “Creating Templates” on page 75).

For smoother rendering of mesh objects, enable mesh smoothing for the entire document in the document preferences (see “Document Display Preferences” on page 60) or for selected meshes on the Render pane of the Object Info palette (see “Applying a Texture to an Object” on page 1530.)

To set the OpenGL options:

1. Select **View > Rendering > OpenGL Options**.

   The OpenGL Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Detail	Specifies the level of detail for renderings; a low setting renders faster

Parameter	Description
Use Textures (Renderworks required)	Renders object textures; this provides better detail and is usually enabled, but it takes longer to render
Use Colors	When selected renders colors, and textures with colors; deselect to render colors as white.
Use Anti-Aliasing	Filters the rendering and reduces pixellated edges; this option is only available if the graphics card supports the feature
Draw Edges (Renderworks required)	Select to render objects with lines drawn around the edges, similar to the lines in the Hidden Line render mode. Edges are drawn where objects meet; break up an object into multiple objects if more edges are needed.
Crease Angle (Renderworks required)	If **Draw Edges** is selected, surfaces that diverge by more than this angle will show crease edges. Set the value to zero to disable crease edges. By default the crease angle is set to 70 degrees.
Use Shadows (Renderworks required)	Objects in the drawing cast shadows on other objects (but not on the active layer plane)
On Ground Only (Renderworks required)	If **Use Shadows** is selected, select this option for drawing objects to cast shadows on a plane at the lowest point in the drawing — which is usually the active layer plane — but not on other objects
Quality (Renderworks required)	If **Use Shadows** is selected, select the level of quality for the shadow edges (a low setting uses less memory and renders faster)

2. Select the OpenGL render options, and click **OK** to change the settings for the file.

Renderworks Features for OpenGL

If the Renderworks product is installed, the OpenGL render mode has additional features that help you to preview and adjust a scene before it is rendered with Renderworks product render modes.

- Turn on the **Draw Edges** feature to outline objects more clearly in the rendered drawing; these edges are similar to those in the Hidden Line render mode.
- Turn on the interactive **Use Shadows** feature to see how the shadows fall with different types of lighting and adjust the lights. When the system hardware and video card are sufficiently powerful, enhanced, highly accurate shadows are used for the OpenGL rendering. This option can be turned off if quality or instability problems are encountered; see “3D Preferences” on page 54.
- OpenGL can display three types of Renderworks backgrounds: One Color, Two Color, and Image. It cannot display Cloud, HDRI, or physical sky backgrounds. (See “Creating Layer Backgrounds” on page 1522 for details.)
- OpenGL is used during viewing operations such as flyovers and walkthroughs when a Renderworks mode is in effect. OpenGL also previews a rendered scene before a Renderworks render has completed.

Rendering with Vectorworks
Rendering with Renderworks

Line Render Options

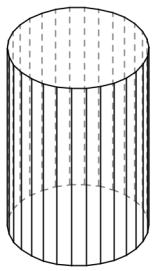
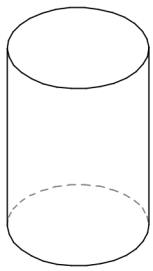
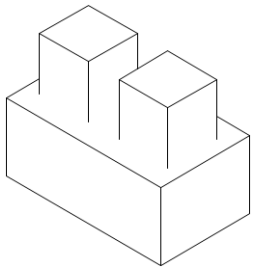
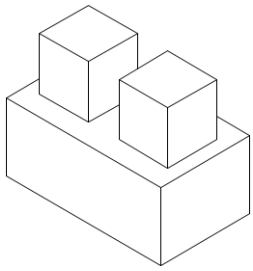
Use the line render options to control the appearance of lines in Hidden Line, Dashed Hidden Line, and Final Shaded Polygon modes. These settings apply only to the current drawing; they remain in effect in the current drawing until the settings are changed. The current line render settings are saved when you create a template (see “Creating Templates” on page 75).

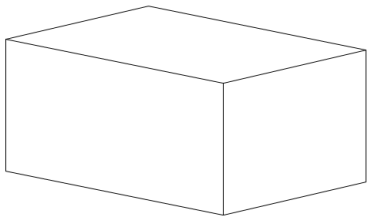
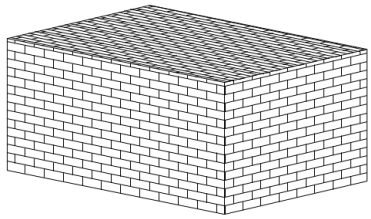
To set line rendering options:

1. Select **View > Rendering > Line Render Options**.

   The Line Render Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Dash Style	Select the dash style for hidden lines when the Dashed Hidden Line mode is selected.
Dash Shade	Adjust the slider to select how hidden lines are shaded when the Dashed Hidden Line mode is selected
Preview	Displays a preview of the selected parameters
Smoothing Angle	Sets the angle to reduce facet lines when the Hidden Line, Dashed Hidden Line, or Final Shaded Polygon mode is selected; if the value is greater than zero, facet lines will be removed between any two faces of an object that are within that angle of each other  A Dashed Hidden Line rendering with the Smoothing Angle set to 0  A Dashed Hidden Line rendering with the Smoothing Angle set to 15
Generate Intersecting Lines	Generates lines where surfaces in the drawing intersect each other when the Hidden Line, Dashed Hidden Line, or Final Shaded Polygon mode is selected; for large files with a lot of curved surfaces, this option can make rendering slower  A Hidden Line rendering with Generate Intersecting Lines disabled  A Hidden Line rendering with Generate Intersecting Lines enabled

Parameter	Description
Display Surface Hatches (Renderworks required)	Select to display surface hatches in a hidden line rendered view  A Hidden Line rendering with **Display Surface Hatches** disabled  A Hidden Line rendering with **Display Surface Hatches** enabled (and brick texture applied)
Display Text and Markers	Select to display text and markers in the rendered view. Hiding text and markers can significantly decrease rendering times.
Sketch Hidden Line Results (Vectorworks Design Series required)	Specifies whether to apply sketch effects to lines when the Hidden Line, Dashed Hidden Line, or Final Shaded Polygon mode is selected; see “Sketch Rendering” on page 1497
Document Default Sketch Style (Vectorworks Design Series required)	Sets the default sketch style for lines when the Hidden Line, Dashed Hidden Line, or Final Shaded Polygon mode is selected; see “Setting the Default Sketch Style” on page 1497. Click **Edit** to open the Sketch Style Editor dialog box for additional sketch style modification.

2. Select the line render options, and click **OK** to change the settings for the file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Rendering with Vectorworks

Rendering with Renderworks

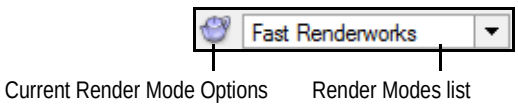
Renderworks Rendering Modes

The Renderworks product offers several rendering modes, including custom rendering options and an artistic rendering option.

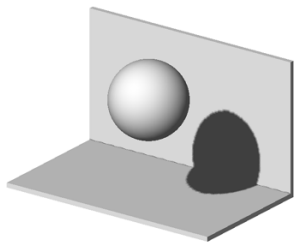
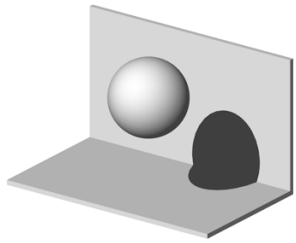
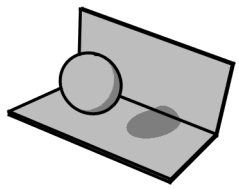
To render a drawing, select **View > Rendering** and the desired rendering mode. The rendering speed depends on the rendering option selected and the number and complexity of objects in the file.

To cancel a rendering before it is complete, press the Esc key.

The View bar displays the **Render Modes** for quick access to the rendering commands. The View bar options display the Render Mode - Short (displaying a shortened version) or the Render Mode - Long (displaying the current render mode options and the current render mode). The **Current Render Mode Options** button opens the settings dialog box for the current render mode, if applicable.



Save files in wireframe mode rather than in a rendered mode, so that files draw more quickly when opened. Rendered viewports are saved in a rendered state when **Save viewport cache** is selected in the Display tab of document preferences.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Rendering Mode	Description
Fast Renderworks	Renders without anti-aliasing or ray tracing; low detail level 
Final Quality Renderworks	Renders with reflections, shadows, anti-aliasing, transparency, and a high level of detail 
Renderworks Style	Renders with parameters that have been saved as a Renderworks Style
Custom Renderworks	Renders with parameters set by the user
Artistic Renderworks	Renders with multiple sketch styles for a hand-drawn look 

For smoother rendering of mesh objects, enable mesh smoothing for the entire document in the document preferences (see “Document Display Preferences” on page 60) or for selected meshes on the Render pane of the Object Info palette (see “Applying a Texture to an Object” on page 1530.)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Background Rendering

Sheet layer viewports and bitmaps created with the **Render Bitmap** tool can render in the background for Renderworks render modes, while you continue working. A sheet layer that requires updating because it is out of date displays with a striped border (see “Creating Sheet Layer Viewports” on page 1616 and “Viewport Status” on
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

page 1664). The viewport renders with the render mode and settings specified for the foreground and background render in the Object Info palette. A render bitmap renders with the tool preference settings in effect when it is created (see “Rendering a Selected Area” on page 1608).

Starting a Background Render

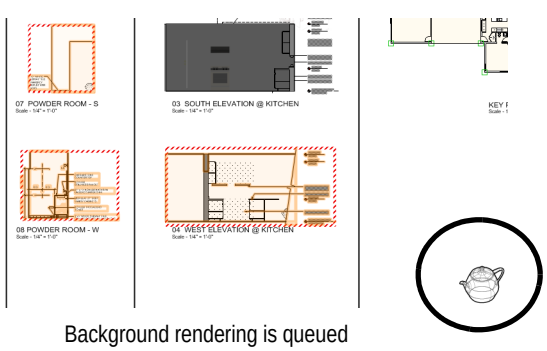
A render bitmap begins rendering in the background as soon as it is created. There are several different ways to update the rendering of sheet layer viewports:

- Select **View > Update Selected Viewports**, or **View > Update All Viewports**.
- Select **Update** from the Object Info palette of selected viewports.
- Right-click (Windows) or Ctrl-click (Mac) on the viewport, and select **Update** from the context menu.
- With one or more sheet layer viewports selected from the Viewports tab of the Navigation palette (Vectorworks Design Series required), select **Update** from the **Navigation** menu. See “The Navigation Palette” on page 199.

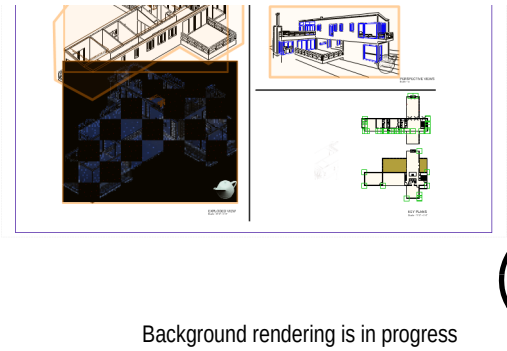
When rendering begins, the Vectorworks program processes the drawing geometry first, as indicated in the message bar. If multiple viewports are selected, the geometry of each one is processed first before background rendering starts. During this time, actions other than canceling the render are blocked. Once the geometry has been processed, background rendering begins and the Vectorworks program is available once again.

> If edits occur to a design layer that affect a current or queued background render of a sheet layer viewport, the render continues, but the viewport will be out of date when the render is complete.

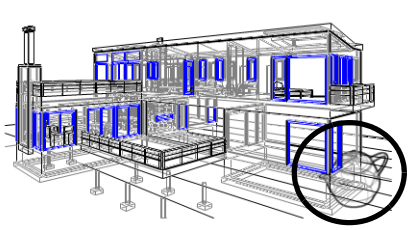
An animated icon at the bottom right of the drawing window indicates that background rendering is queued or is in progress. This icon displays even when you switch to another layer, as a reminder that background rendering is occurring. Another icon, in the bottom right corner of the viewport or render bitmap, indicates the individual status of each viewport or bitmap. The Viewports tab of the Navigation palette (Vectorworks Design Series required) and message bar also display information about the rendering status. During background rendering, the rendering viewports and bitmaps are locked, and only the **Cancel Update** option is available from the Object Info palette; however, all other drawing processes are available, including switching to another layer, editing objects, and even moving render bitmaps to another layer. Printing, saving, exporting images, or closing the Vectorworks program when background rendering is in process will generate an alert message with the option to proceed with, or cancel, the rendering.



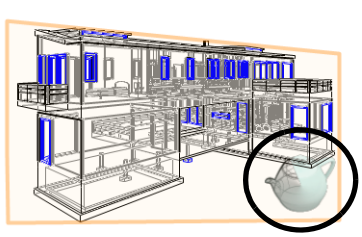
Background rendering is queued



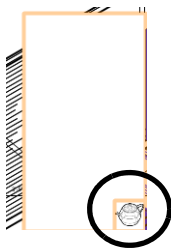
Background rendering is in progress



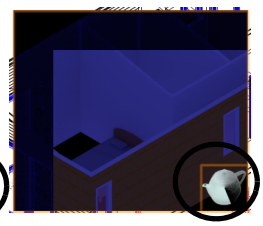
Sheet layer viewport is in rendering queue



Sheet layer viewport rendering is in progress



Render bitmap is in rendering queue



Render bitmap rendering is in progress

The message bar indicates the name of each sheet layer viewport as it is rendered. When the individual and overall background render is complete, the icon no longer displays.

Canceling a Background Render

There are several different ways to cancel sheet layer viewports and render bitmaps currently being rendered in the background or waiting in a render queue.

- Select **View > Cancel All Viewport Updates**.
- Select **Cancel Update** from the Object Info palette of selected viewports, or **Cancel Rendering** to cancel the rendering of selected render bitmaps.
- With one or more sheet layer viewports selected from the Viewports tab of the Navigation palette (Vectorworks Design Series required), select **Cancel Update** from the **Navigation** menu. See “The Navigation Palette” on page 199.

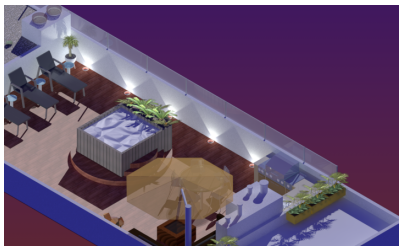
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Rendering a Selected Area
Updating Viewports
Viewport Status
The Navigation Palette
Rendering with Renderworks
Optimizing Rendering Performance

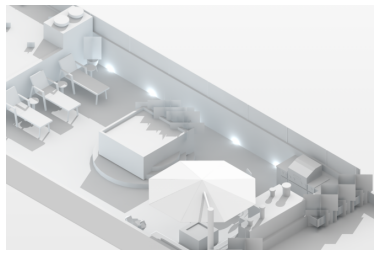
Renderworks Styles

Several elements combine to create a satisfactory rendering: the selected render mode, the render mode options, the lighting options, and the selected Renderworks background. A Renderworks style saves the settings for these parameters as a resource that can be re-applied later and shared between files.

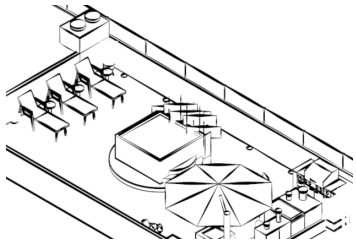
The Renderworks product includes several default styles to quickly obtain a variety of looks, with no parameter adjustment or rendering knowledge required.



Realistic Exterior Night Final



Realistic Colors White



Artistic Taper Thick Black

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Renderworks Styles
Applying Renderworks Styles
Rendering with Renderworks

Creating Renderworks Styles

Renderworks styles that incorporate either realistic or artistic Renderworks parameters can be saved as resources. For more information on creating resources, see “Creating New Resources” on page 228.

If you create a Renderworks style while the layer or viewport is rendered in any Renderworks mode or a Renderworks style, the current settings are used as the basis for the new style. This is a convenient way of saving render settings.

To create a Renderworks Style resource:

1. Select **Window > Palettes > Resource Browser** to open the Resource Browser.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Renderworks Style**.

   The Edit Renderworks Style dialog box opens. Provide a name for the style resource and select a realistic or artistic type of style, and then specify the render parameters.

Click to show/hide the parameters.

Parameter	Description
Name	Specify the name of the Renderworks Style resource
Type	Select either Realistic or Artistic; different options are available depending on the desired look
Realistic	Creates a Renderworks style based on Custom Renderworks options
Options	
Anti-Aliasing	Select for smoother edges on objects and textures; deselect for faster rendering with rougher edges
Shadows	Render shadows for a higher degree of realism
Blurriness	Select to render textures with blurred reflectivity and/or blurred transparency (this can add significant rendering time)
Textures	Renders the textures assigned to objects; deselect for faster rendering
Displacement Mapping	Renders with displacement mapping when a texture's bump shader setting has a displacement height set; select the displacement mapping quality on the Quality tab. Deselect for faster rendering.
Colors	When selected, renders colors, and textures with colors; deselect to render colors as white
Image Exposure (%)	Adjusts the exposure brightness of the rendered image; enter a value to increase the brightness (above 100%) or to decrease the brightness (below 100%)
Quality	For each parameter, select the quality level. Higher quality results in better resolution of rendered images, with better texture detail and softer shadows, but takes more time.
Quality Levels	Conveniently sets the quality level of all the options at one time. Custom indicates that some options have different quality levels set.
Curved Geometry	Select the quality level for faceting of curved geometry (such as NURBS surfaces)
Anti-Aliasing	Select the quality level for anti-aliasing (smoothing) of edges on objects and textures
Indirect Lighting	Select the quality level for indirect lighting effects
Soft Shadows	Sets the quality level of shadows for light objects that have **Soft Shadows** enabled. For area and line lights, specifies the sampling quality of the light.
Blurriness	Select the quality level of textures with blurry reflectivity and/or transparency
Environment Lighting	Select the sampling quality level for environment (HDRI) background lighting. This option has no effect when indirect lighting is enabled for the lighting options.

Parameter	Description
Displacement Mapping	When enabled on the Options tab and set for a bump shader, select the quality of the displacement mapping. Higher quality is more realistic for fine displacement bumps, but will require longer rendering times. Rendering can be significantly slower with displacement mapping.
Max Reflections	Enter the number of levels of reflection among shiny surfaces; a higher value slows rendering, but can yield a more realistic image for scenes with many inter-reflecting objects
Lighting	
Apply Lighting Options	Controls whether the Renderworks style changes the current lighting options when it is applied. Select the option to apply the style's lighting options to the layer or viewport when the Renderworks style is applied. Deselect the option to leave the current lighting options unchanged when the Renderworks style is applied.
Lighting Options	Sets the lighting options; see "Setting Lighting Options" on page 1571
Edit Resource	Opens the Edit Renderworks Background dialog box, to conveniently change background and environment lighting options that affect the rendering
Edges	
Apply Artistic Edges	Select to apply artistic edges on top of the realistic shaded rendering
Style	Select an edge style
Thickness (pixels)	Specifies the edge thickness
Color	Specifies the edge color
Background	
Apply Renderworks Background	Controls whether the Renderworks Background changes the current background options when it is applied. Select the option to apply the style's background to the layer or viewport when the Renderworks style is applied. Deselect the option to leave the current background unchanged when the Renderworks style is applied.
Renderworks Background	Sets the Renderworks Background options; see "Creating a Background" on page 1522
Artistic	Creates a Renderworks style based on Artistic Renderworks options
Artistic Style	Select an Artistic Renderworks style and set any options
Background	
Apply Renderworks Background	Controls whether the Renderworks Background changes the current background options when it is applied. Select the option to apply the style's background to the layer or viewport when the Renderworks style is applied. Deselect the option to leave the current background unchanged when the Renderworks style is applied.
Renderworks Background	Sets the Renderworks Background options; see "Creating a Background" on page 1522

4. Click **OK** to create a Renderworks Style resource with the name and properties specified in the Edit Renderworks Style dialog box. The Renderworks style displays in the Resource Browser and is saved with the file. It is added to the list of available Renderworks styles in **View > Renderworks Style**.

To edit a Renderworks Style resource after it has been created, select the Renderworks style from the Resource Browser and select **Resources > Edit**. See “Working with Resources” on page 225.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Applying Renderworks Styles
Custom Renderworks Options
Artistic Renderworks Options
Renderworks Styles
Using the Resource Browser

Applying Renderworks Styles

Renderworks styles are resources that are saved with the file and can be found in the Resource Browser. The Vectorworks program comes with several Renderworks styles that are provided as default content (default content is automatically imported into the current file at the point of use and displays in the Resource Browser; see “Resource Libraries” on page 219).

To render a drawing with a Renderworks style:

Select **View > Renderworks Style**, and then the desired style from either the default content or the current file’s content. The current Renderworks style in effect displays with a check mark on the **Renderworks Style** menu. The rendering speed depends on the rendering option selected and the number and complexity of objects in the file. To cancel a rendering before it is complete, press the Esc key.

A Renderworks Style resource can also be applied by dragging it from the Resource Browser into a design layer of the drawing.

Renderworks styles are also available from the **Render Mode** menu on the View bar, as described in “Rendering with Renderworks” on page 1593. When creating a viewport, Renderworks styles are available from the **Render** list in the Create Viewport dialog box, or from the Object Info palette of an existing viewport. The **Render Bitmap** tool and batch rendering functionality can also take advantage of Renderworks styles.

When a Renderworks background is set from the Renderworks style, the option to select a background may not be available when the style is in effect, such as for viewports. The Renderworks style’s background settings override the Renderworks background settings when **Apply Renderworks Background** was selected for the style. Similarly, when lighting options have been set from the Renderworks style, any attempt to edit lighting options automatically opens the Edit Renderworks Style dialog box, since these are the settings in effect when **Apply Lighting Options** is selected for the style.

Use Renderworks styles and the **Heliodon** tool (Vectorworks Design Series required) to obtain good results from existing files. Open an existing model, and either delete or turn off existing lights in the drawing (through the Visualization palette). In the OpenGL Options, select **Use Shadows**. Render with the OpenGL render mode and select a 3D view. (A Renderworks Camera object can help to set and adjust the 3D view.)

Place a heliodon object in the drawing with the **Heliodon** tool. Click **Solar Animation** to interactively adjust the date and time (see “Solar Studies” on page 1489). This is particularly effective with physical sky Renderworks backgrounds.

For exteriors, select the **Realistic Exterior Fast** or **Realistic Exterior Final** Renderworks style; for interiors, select either **Realistic Interior Fast** or **Realistic Interior Final**. The heliodon creates a strong sun light with soft-edged shadows, while the Renderworks style provides soft sky lighting from the included HDRI backgrounds.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Renderworks Styles
Setting Lighting Options
Custom Renderworks Options
Artistic Renderworks Options

Renderworks Styles
Using the Resource Browser
Solar Studies

Custom Renderworks Options

Custom render settings provide greater control over fine-quality rendering. These settings are used exclusively by the **Custom Renderworks** rendering mode. Custom geometry settings override the **3D conversion resolution** setting on the 3D tab in Vectorworks preferences. These settings apply only to the current drawing and remain in effect in the current drawing until the settings are changed. In addition, the settings are saved when creating templates (see “Creating Templates” on page 75).

To set Custom Renderworks rendering options:

1. Select **View > Rendering > Custom Renderworks Options**.

   The Custom Renderworks Options dialog box opens. The default options match Fast Renderworks rendering mode.

Click to show/hide the parameters.

Parameter	Description
Options	Enabling the options allows for better quality, higher detailed rendering, but takes longer to render
Anti-Aliasing	Select for smoother edges on objects and textures; deselect for faster rendering with rougher edges
Shadows	Render shadows for a higher degree of realism
Blurriness	Select to render textures with blurred reflectivity and/or blurred transparency (this can add significant rendering time)
Textures	Renders the textures assigned to objects; deselect for faster rendering
Displacement Mapping	Renders with displacement mapping when a texture’s bump shader setting has a displacement height set; select the displacement mapping quality below. Deselect for faster rendering.
Colors	When selected, renders colors, and textures with colors; deselect to render colors as white
Lighting Options	Click **Layer Lighting Options** (from a design layer) or **Viewport Lighting Options** (from a viewport) to quickly access the associated lighting options dialog box.
Image Exposure (%)	Adjusts the exposure brightness of the rendered image; enter a value to increase the brightness (above 100%) or to decrease the brightness (below 100%)
Quality	For each parameter, select the quality level. Higher quality results in better resolution of rendered images, with better texture detail and softer shadows, but takes more time.
Quality Levels	Conveniently sets the quality level of all the options at one time. Custom indicates that some options have different quality levels set.
Curved Geometry	Select the quality level for faceting of curved geometry (such as NURBS surfaces)
Anti-Aliasing	Select the quality level for anti-aliasing (smoothing) of edges on objects and textures
Indirect Lighting	Select the quality level for indirect lighting effects

Parameter	Description
Soft Shadows	Sets the quality level of shadows for light objects that have **Soft Shadows** enabled. For area and line lights, specifies the sampling quality of the light.
Blurriness	Select the quality level of textures with blurry reflectivity and/or transparency
Environment Lighting	Select the sampling quality level for environment (HDRI) background lighting. This option has no effect when indirect lighting is enabled for the lighting options.
Displacement Mapping	When enabled in the **Options** and set for a bump shader, select the quality of the displacement mapping. Higher quality is more realistic for fine displacement bumps, but will require longer rendering times. Rendering can be significantly slower with displacement mapping.
Max Reflections	Enter the number of levels of reflection among shiny surfaces; a higher value slows rendering, but can yield a more realistic image for scenes with many inter-reflecting objects

When testing a rendering, set the **Quality** levels to low and turn **Anti-Aliasing** and even **Shadows** off. Lower rendering quality is offset by faster rendering times.

To create a "white card" or white model rendering, turn off textures and colors.

2. Click **OK** to set the Custom Render options.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Renderworks Rendering Modes
Setting Lighting Options

Artistic Renderworks Options

When you select the Artistic Renderworks mode, a variety of styles and options are available to create a hand-drawn or artistic look for a 3D drawing. These settings apply only to the current drawing and remain in effect in the current drawing until the settings are changed. In addition, the settings are saved when creating templates (see "Creating Templates" on page 75).

Artistic Renderworks options are available as part of a Renderworks style; see "Renderworks Styles" on page 1596. A Renderworks style allows artistic parameters to be saved as a resource to be re-applied later and shared between files.

Artistic Renderworks does not produce sketch rendering of 2D objects.

To set Artistic Renderworks options:

1. Select **View > Rendering > Artistic Renderworks Options**.
2. The Artistic Renderworks Options dialog box opens. A preview scene is shown in order to evaluate the effects of the different options.
3. Select the **Style** from the list, and then click **Options** to set specific style parameters. The preview image displays the resulting effect.

   Most Artistic Renderworks styles use hidden line rendering (except for Art and Hatch). These styles may require longer rendering times for drawings with many facets (polygons). A style that does not use hidden line rendering is recommended for extremely complex drawings.

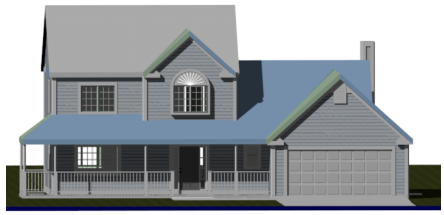
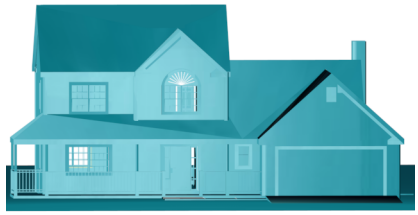
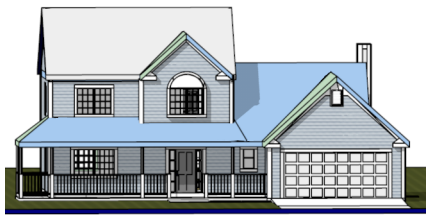
   The Artistic Renderworks styles allow the layer's Renderworks background to show through, if one has been defined and applied (see "Applying Renderworks Backgrounds" on page 1545).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

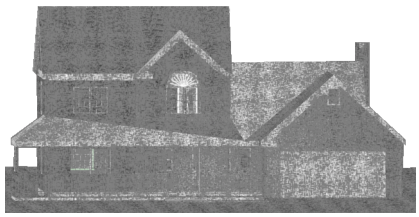
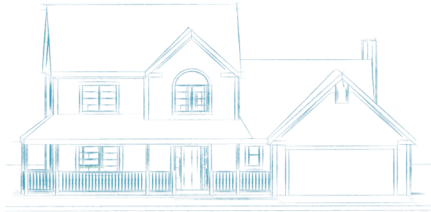
Many of the artistic styles share the parameters of edge color and edge thickness. Some parameters are uniquely available to certain styles.

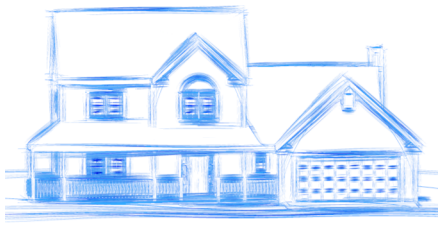
Click to show/hide the parameters.

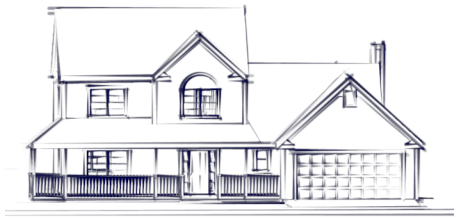
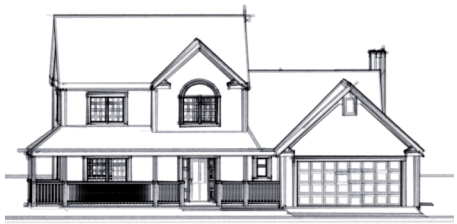
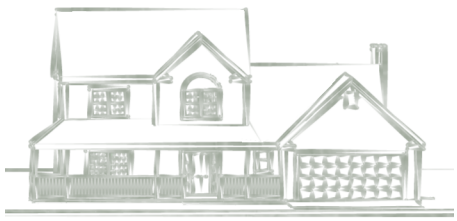
Parameter	Description	Style Where Applied
Edge Thickness	Specifies the thickness of lines used in the style; for lines that vary in thickness, this sets the thickest portion of the line	All styles except Art and Hatch
Edge Color	Click the color box to select the color of the lines	All styles except Art and Hatch
Shadow Color	Click the color box to select the color of shadows	Lines and Shadow
Change Image	Selects a different image to apply to Art mode; the image selected must be square, and contain a color-mapped sphere. The color of the sphere is used to color the image.	Art

Click to show/hide the options.

Artistic Renderworks Option	Description
None	For comparison, the image below is rendered with Final Quality Renderworks 
Art	Transforms the image into a single color with shadows 
Cartoon	Traces a border around edges of objects and combines simplified shading with drawn edges 

Artistic Renderworks Option	Description
Hatch	Shades the image with stroked hatch lines 
Lines and Shadow	Traces a border around the edges of objects and displays shadows 
Pencil (soft)	Simulates a hand-drawn pencil drawing with soft, smudged pencil line(s) at the edges of objects 
Pencil (Sketch Concept)	Simulates a hand-drawn pencil drawing with thin lines that extend slightly past their intended point 
Pencil (Rough Sketch)	Simulates a hand-drawn pencil drawing with many overlapping lines 

Artistic Renderworks Option	Description
Pencil (Loose)	Simulates a hand-drawn pencil drawing with curvy, careless straight lines 
Pen (Thick and Thin)	Simulates a hand-drawn pen drawing with uneven pressure on the pen nib 
Pen (Thin Distance)	Simulates a hand-drawn pen drawing with lines that get thinner as the line retreats 
Crayon	Draws edges with a thick, monochrome crayon 
Chunky Lines	Draws lines with a hesitant stroke that simulates the ink sinking into the paper at those points 

Artistic Renderworks Option	Description
Cloned Lines	Draws overlapping, tapering lines 
Brush (Watercolor)	Draws as with a thin watercolor brush, with ink that soaks into the paper 
Brush (Watercolor Light)	Similar to Brush (Watercolor), but with less pressure applied 
Brush (Chinese)	Draws as with ink and a watercolor brush, with thicker lines at the corners 
Brush (Chinese Fade)	Draws as with ink and a watercolor brush, with lines that start thick and become thinner along the stroke 

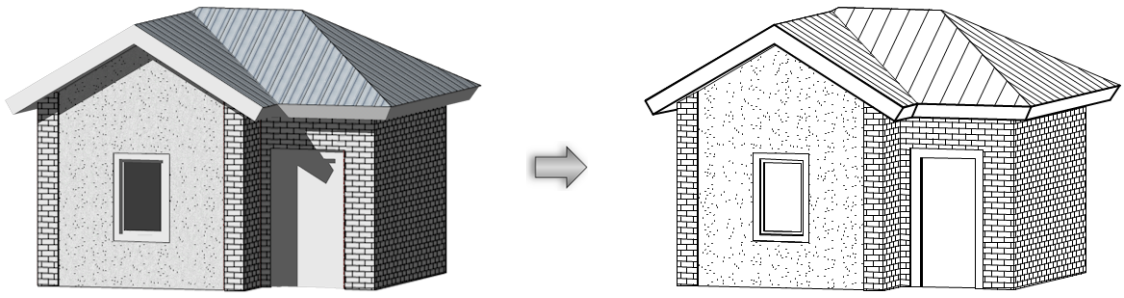
4. Click **OK** to set the Artistic Renderworks options.

5. Select **View > Rendering > Artistic Renderworks** to render with the specified option.

Applying Colors

Hidden Line Rendering with Surface Hatches

In hidden line render mode, it is possible to produce monochromatic conceptual or production drawings by displaying planar hatches wherever a textured surface would exist in a raster-rending mode (Renderworks required).



Drawing with texture

Drawing with hidden line rendering surface hatches

Surface hatches are an aspect or feature of a texture; they can be used when associated with the texture that is applied to an object. Some of the most commonly used Renderworks standard textures are pre-associated with an appropriate hatch, but the surface hatches can be associated with other textures, changed, and settings can be edited as desired.

> Most of the textures with pre-associated surface hatches are material textures that would be applied to building external surfaces, such as brickwork, CMU, concrete, corrugated metal, etc. They are easily identifiable in texture libraries by the text <surf hatch> in the resource name.
>
> If a hatch that is associated with one or more textures is deleted, all textures associated with that hatch have their hatch association set to None.

To display surface hatches on a design layer using an existing hatch and texture association:

1. Select **View > Rendering > Line Render Options**.

   The Line Render Options dialog box opens.
2. Select **Display Surface Hatches** and click **OK**.

To display surface hatches in a viewport using an existing hatch and texture association:

1. On the viewport's Object Info palette Shape tab, set **Background Render** and/or **Foreground Render** to Hidden Line.
2. Click **Background/Foreground Render Settings** to open the Hidden Line Render Settings dialog box.
3. Select **Display Surface Hatches** and click **OK**.

   Click here for a video tip about this topic (Internet access required).

Editing Surface Hatches

To associate a hatch with a texture or to edit an existing hatch and texture association:

1. From the **Resource Browser**, select the texture, and then select **Resources > Edit**.

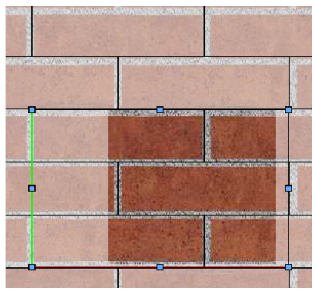
   The Edit Texture dialog box opens.
2. **Surface Hatch** displays the name of the surface hatch resource associated with the texture, if any. To associate a new hatch or align and register the existing hatch with the texture, select **Edit Surface Hatch**.

The Edit Surface Hatch dialog box opens.

To navigate directly to the Edit Surface Hatch dialog box without opening the Edit Texture dialog box, select **Edit Surface Hatch** from the pull-down menu to the right of the **Texture** list on the Object Info palette Render tab.

Hatches defined with page units cannot be aligned to a texture; therefore the alignment fields are grayed out in the Edit Surface Hatch dialog box.

Click to show/hide the parameters.

Parameter	Description
Surface Hatch	From the list of hatches present in the drawing, select a surface hatch to associate with the texture, or select None
Set Hatch Alignment	
X/Y Offset	Sets the horizontal/vertical surface hatch offset relative to the texture
I/J Length	Sets the horizontal/vertical repeat distance relative to the hatch definition (I in the red direction and J in the green direction)
Lock button	Select to maintain the aspect ratio between the I and J lengths
Rotation	Sets the degree of rotation from horizontal relative to the texture
Flip Horizontally/ Vertically	Flips surface hatch horizontally/vertically relative to the texture
Reset	Reset surface hatch alignment settings
Preview	Displays an outline of the hatch over an image of the texture; control points can be moved for manual adjustment of alignment and registration 
Preview Controls	Adjusts the preview position and magnification. Click **Pan** and drag the preview to the desired location. Click **Zoom In** or **Zoom Out** and then click and drag to create a marquee; this zooms in or out on a particular section of the preview. Click **Fit** to fit the preview to the window (according to **Obj Size**).
Texture Size	Sets the texture size. This field corresponds to the **Size** parameter in the Edit Texture dialog box; changes in one location affects the other.

3. Set the desired parameters to align and register the hatch with the texture, and click **OK** to return to the Edit Texture dialog box.
4. Click **OK**.

Like textures, surface hatches can be applied to an object by class; see “Applying Object Textures by Class” on page 1542.

Line Render Options
Creating a New Texture
Applying a Texture to an Object

Rendering a Selected Area

The **Render Bitmap** tool creates a rendered bitmap image of a selected area and places it on top of the drawing. Manipulate this image similar to imported images.

Use this tool to preview a section of a drawing, or create a layout of several rendered views. It is also possible to render the entire drawing, creating an image that can be sent to a printer for preview or final quality output, or exported into a different image file format and saved.

Like sheet layer viewports, render bitmap images can render in the background for Renderworks render modes, while you continue working in the drawing. See “Background Rendering” on page 1594.

The **Render Bitmap** tool renders the image with the specified resolution and rendering mode. The bitmap created by the tool draws using the options selected on the Resolution tab in the document preferences (see “Resolution Preferences” on page 62). Choose **Full Resolution**, **Reduced Resolution** or **Bounding Box**. The bounding box option displays as a gray rectangle bitmap image. Using this option saves re-drawing time when using the **Pan** tool or scroll bars.

The image created by the **Render Bitmap** tool is compressed by either the JPEG or PNG compression method, to reduce file size. The compression used depends on the **Default compression** selected on the Edit tab of Vectorworks preferences (see “Edit Preferences” on page 49).

To render a specified area:

1. Select the **Render Bitmap** tool from the Visualization tool set.
2. Click the **Preferences** Tool bar button.

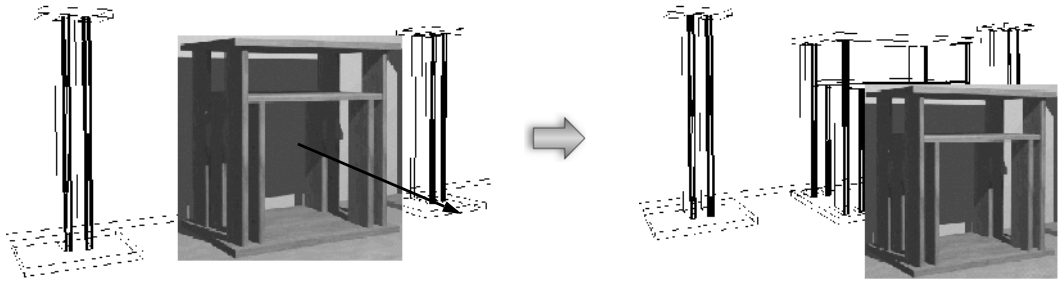
   The Render Bitmap Settings dialog box opens. Specify the rendering mode and any options.

Click to show/hide the parameters.

Parameter	Description
Render Mode	Select the rendering mode for the image; if the mode has additional settings, click **Options** to make any changes (any changes also apply to the design layer settings)
Resolution (DPI)	Specify the resolution for the bitmap in dots per inch, relative to the page; lower resolution values reduce file size

3. Click **OK**.
4. Click and drag to create a marquee box around the desired area.

   The area is rendered (using background rendering for any Renderworks modes). Cancel the rendering by clicking **Cancel Rendering** from the Shape tab of the Object Info palette. When complete, a bitmap image of the rendered area is placed on top of the original area.
5. Move the new image to the desired location.



After rendering, the rendered bitmap image can be cut from the Vectorworks drawing and pasted into any image editing program for further manipulation.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Background Rendering
Rendering with Vectorworks
Rendering with Renderworks

Batch Rendering

Batch rendering allows several images to be rendered while the computer is unattended. Each batch job retains its own view, rendering mode, resolution, and export settings.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Batch Rendering Jobs
Starting the Rendering Job Batch

Creating Batch Rendering Jobs

The batch rendering job specifies the job name, rendering mode and options, and image dimensions and format, for the current drawing file.

Batch rendering does not retain geometry or lighting information; it saves the view, render, and export settings. To export different lighting situations for a scene, create sheet layer viewports and change the lighting with the Visualization palette (see "Managing Lights and Cameras with the Visualization Palette" on page 1611). Viewports allow lighting overrides without affecting the lighting on the design layer. This method allows batch rendering of viewports of the same scene with different lighting conditions.

To create a batch rendering job:

1. Establish the desired file view and level of magnification.

   The Export Image File settings determine the specific dimensions of the exported area and the visibility of objects (see "Exporting an Image File" on page 1683).

2. Select **View > Rendering > Create Batch Render Job**.

   The Create Batch Render Job dialog box opens. Specify the rendering job parameters and click **OK**.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies the rendering job name for creating the batch
Render Mode	Select the rendering mode from the list

Parameter	Description
Options	For **OpenGL** and **Custom Renderworks** rendering modes, the default file settings are in effect; to change the settings, click **Options**. These changes apply to the current job only. For more information on OpenGL settings, see "Rendering with Vectorworks" on page 1587. For Custom Renderworks settings, see "Custom Renderworks Options" on page 1600.
Set Export Image File Options	Opens the Export Image File dialog box for specifying the rendered image file settings, including export area and format

The **Marquee** export area option in the Export Image File dialog box is not available for batch rendering jobs.

3. Create additional batch rendering jobs as required.

   Batch jobs can be renamed, edited, or deleted from the Start Batch Render dialog box.

Starting the Rendering Job Batch

Once rendering jobs have been created, specify their order and the rendered file location.

To specify the rendering job batch:

1. When it is time to begin the batch rendering process, select **View > Rendering > Start Batch Render**.

   The Start Batch Render dialog box opens. Select the jobs to be rendered and specify the file location for the resulting rendered images.

Click to show/hide the parameters.

Parameter	Description
Available Jobs	Lists the current batch render jobs in order of job creation; the jobs are displayed with the image file extension to be generated according to the job settings
Chosen Jobs	Lists the currently selected render jobs in order of execution; the most recently added job is placed at the end of the list
Choose All	Moves all **Available Jobs** to the **Chosen Jobs** list
>> button	Moves the selected job from the **Available Jobs** list to the end of the **Chosen Jobs** list
<< button	Moves the selected job from the **Chosen Jobs** list to the end of the **Available Jobs** list
Duplicate	Copies the selected available job and adds it to the end of the **Available Jobs** list; specify a new name for the job in the Assign Name dialog box
Edit	Opens the Edit Batch Render Job dialog box, for changing the parameters of the selected job
Delete	Removes the selected job from the list of available render jobs (no undo)
Browse (Windows)/ Choose (Mac)	Specifies the **Parent Folder** location, where all batch rendering resulting files will be located. By default, this is the application folder. The result subfolder name is also displayed (this subfolder is named according to the current date and time).

2. Click **Start** to begin the batch rendering.

   The Batch Render Progress dialog box displays the status of the batch rendering. Press the Esc key to cancel the current job and proceed with the remaining batch jobs. Press **Cancel** to cancel all batch rendering jobs.

3. As each job is rendered, the resulting file is placed in the Results Folder.

Managing Lights and Cameras with the Visualization Palette

The Visualization palette provides fast and easy access to all lights and cameras in a Vectorworks file. Lights and cameras can be selected, edited, duplicated, created, and deleted. The palette also controls lighting overrides for selected sheet layer viewport(s), without modifying the lighting on the design layer or in another viewport.

Tasks that can be performed from the Visualization palette include:

- Quickly locating any light in the file;
- Turning lights on and off;
- Editing of any light or lights in the file, even lights that are inside a container object or on a different layer;
- Creating lighting overrides for sheet layer viewports;
- Managing light and camera objects, including creating, editing, deleting, duplicating, and sorting;
- Quickly switching to a camera's established view;
- Turning heliodon objects on or off to conduct sun studies (Vectorworks Design Series required).

To use the Visualization palette:

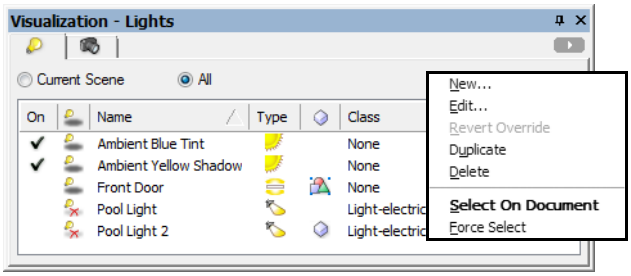
1. Select **Windows > Palettes > Visualization**.

   The Visualization palette opens. The Visualization palette can remain open while you work in the drawing.

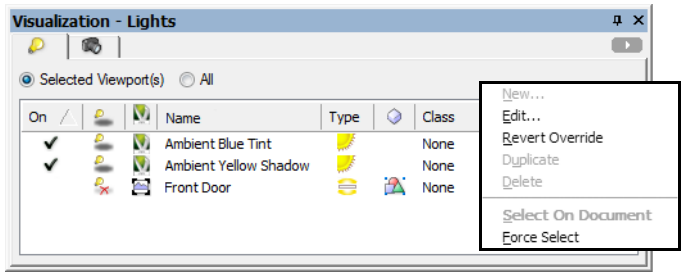
2. The Visualization palette contains two tabs: the Lights tab accesses lights, and the Cameras tab accesses cameras. The lights and cameras can be sorted by clicking in the header of any column.

   The Lights tab functions in two different modes depending on whether you are on a design layer or sheet layer. On a design layer, the palette lists either all the lights in the file, or just the lights that affect the current view. On a sheet layer, the palette lists either all the lights in the file, or just the lights of selected viewports; edits to the selected viewport(s) affect the viewport lighting, creating an override, but this does not affect the associated design layer lighting.

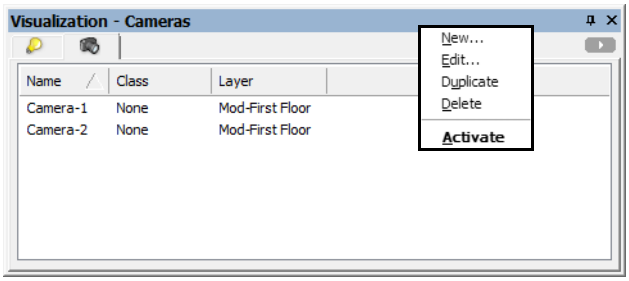
   Depending on what action is being performed, different options are available in both the Visualization palette and the **Visualization palette** menu. To open the **Visualization palette** menu, click on the small triangle at the top right of the palette, or right-click (Windows) or Ctrl-click (Mac) in the palette to open a context menu.



Design layer Visualization palette options



Sheet layer Visualization palette options



Visualization palette camera options

Click to show/hide the parameters.

Parameter	Description
Lights tab	The Lights tab accesses the parameters of sheet layer viewports or all lights in the file
Current Scene	Displays only the lights which affect the current view (lights from the current layer and in visible classes)
Selected Viewport(s)	Displays only the lights which affect the currently selected sheet layer viewport(s); in this mode, lights can be edited to override the lighting of the selected viewports only. (Other Visualization palette light management commands are not available.) Select **Revert Override** to return the viewport to its original lighting.
All	Lists all the lights in the file
On	Click to toggle the light on or off; a check mark indicates that the light is on
(Cast Shadows) 	Click to toggle the ability of the lights to cast shadows on or off; a red "x" indicates that shadows are off
(Override State) 	For lights in selected viewport(s), indicates whether the light has an override. •  Uses design layer definition: the light parameters are the same in both the design layer and the viewport(s). •  Uses viewport override definition: the light parameters have been edited for the selected viewport(s), creating a viewport lighting override.
Name	Displays the name of the light (as shown on the Data tab of the Object Info palette)
Type	Indicates the type of light: •  Directional •  Point •  Spot •  Custom • Area • Line
(Light Parent) 	Indicates whether the light is within a container object; if blank, the light is not part of another object. •  Group: indicates that the light is contained within a group. •  Symbol: indicates that the light is contained within a symbol; each instance is listed in the palette. Edits made directly from the Visualization palette, such as turning lights on and off, affect all symbol instances. (Selecting **Visualization > Edit**, however, edits the symbol instance properties.) •  Plug-in Object: indicates that the light is contained in a plug-in object, such as a lighting device (Vectorworks Spotlight required) •  Layer Link: indicates that the light is contained within a layer link
Class/Layer	Displays the light's class and layer
Menu commands	The **Visualization palette** menu commands manage and select light objects

Parameter	Description
New	Opens the Create Light dialog box, to create a directional, point, spot, or custom light on the current design layer; see “Adding Light Sources” on page 1574
Edit	Opens the Properties dialog box, to edit the parameters of the currently selected light(s); when more than one light is selected, only common properties can be edited
Revert Override	Restores the original light parameters for the viewport(s), when the lights in selected sheet layer viewport(s) have been edited
Duplicate	Copies the currently selected light(s), placing the copy or copies on the same layer and in the same class as the original(s). Lights in symbols or plug-in objects cannot be duplicated.
Delete	Deletes the currently selected light(s). Lights in symbols or plug-in objects cannot be deleted.
Select On Document	Selects the current light(s), and automatically centers the view on the light, or its container (alternatively, double-click on a light in the Visualization palette, if the light is in the active class or layer.) The light properties can then be edited in the Object Info palette or by clicking **Visualization > Edit**. The selection and editing of multiple lights at once is supported. If the light is not in the active class or layer, use the **Force Select** command instead. The Vectorworks Display preference must be set to show lights.
Force Select	If the light is in a different class or layer, activates the layer or class, and automatically centers the view on the light, or its container group or symbol. The light properties can then be edited in the Object Info palette or by clicking **Visualization > Edit**. Multiple lights cannot be selected by this method.
Cameras tab	The Cameras tab accesses the parameters of the camera objects in the drawing
Name	Displays the name of the camera (as shown on the Data tab of the Object Info palette)
Class/Layer	Displays the camera’s class and layer
Menu commands	The **Visualization palette** menu commands manage and select camera objects
New	Activates the **Renderworks Camera** tool; click to place a camera object
Edit	Opens the Properties dialog box, to edit the parameters of the currently selected camera(s)
Duplicate	Copies the currently selected camera(s), placing the copy or copies on the same layer and in the same class as the original(s)
Delete	Deletes the currently selected camera(s)
Activate	Selects the current camera, and automatically changes the view to that of the camera (alternatively, double-click on a camera in the Visualization palette)

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Light Source Properties
Setting a Camera View in Renderworks
List Box Functionality
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewports

Viewport objects allow you to create views from several directions, complete with details, annotations, dimensions, and title blocks. Viewports can show other parts of the active document, or even portions of other documents.

Viewports can display entire, as well as cropped, views of a drawing, with specified layer and class visibility settings, projection, render mode, and orientation parameters. If the drawing changes, you can easily update the viewports to reflect the changes.

There are several different types of viewports, depending on whether you have the Vectorworks Fundamentals product or one or more Vectorworks Design Series products.

- Sheet layer viewports

  In both the Vectorworks Fundamentals and Vectorworks Design Series products, you can create one or more viewports on a sheet layer, and each viewport can show one or more design layers from this document. Sheet layer viewports, often created for presentation purposes, are created on special layers called sheet layers. Sheet layers retain their own print settings, including print area, resolution, and printer setup parameters. For more information on sheet layers, see “Layers” on page 161.

  When Renderworks is installed, a sheet layer viewport can be linked to a Renderworks camera.

- Design layer viewports (Vectorworks Design Series required)

  Vectorworks Design Series products allow you to create one or more viewports on a design layer, and the design layers shown in the viewports can be either from this document, or referenced from another document. Like a sheet layer viewport, a design layer viewport can display design layers from the current file in a full or cropped view; unlike a sheet layer viewport, it can include one or more design layers that are referenced from another file.

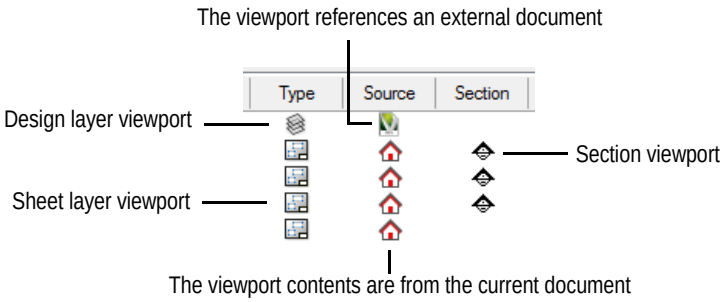
- Section viewports (Vectorworks Design Series required)

  A section viewport creates a cross section view of a model, but leaves the model intact. A section viewport can be created from a design layer or another non-sectioned viewport; it can be placed on a design layer or sheet layer.

- Detail viewports (Vectorworks Design Series required)

  A detail viewport is a cropped sheet layer viewport that shows a detail view of any part of a drawing. A detail viewport can be created from a design layer, another sheet layer viewport, or a section viewport. The crop object used to create the detail viewport becomes a callout object that is linked to the detail viewport for easy navigation between layers.

The Viewports tab of the Organization dialog box differentiates among the various types of viewports.



Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Sheet Layer Viewports
Creating Design Layer Viewports
Creating Section Viewports
Creating Detail Viewports
Viewport Properties
Modifying Viewports
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewport Status
Updating Viewports

Creating Sheet Layer Viewports

Creating a Sheet Layer Viewport from a Design Layer

To create a viewport from a design layer:

1. Select **View > Create Viewport**.
2. The Create Viewport dialog box opens. The viewport parameters are initially set to be the same as those of the design layer that is currently active, but they can be changed here. After the viewport has been created, additional parameters become available; see “Viewport Properties” on page 1636.

   For Vectorworks Design Series products, the Create Viewport dialog box has additional functionality for creating design layer viewports; see “Creating a Design Layer Viewport from an Internal Design Layer” on page 1621.

Click to show/hide the parameters.

Parameter	Description
Viewport Name	Specifies the viewport name; this name must be unique in the document
Drawing Title	Specifies a descriptive title for the sheet layer viewport. This name displays as the drawing title for any annotation objects (drawing labels, section markers, and sheet borders) that are added to the viewport. The drawing title name is limited to 63 characters.
Create on Layer	Select the sheet layer where the viewport will be created, or select New Sheet Layer to create a sheet layer. If there are no sheet layers present and a new one is not created now, you will be prompted to create a sheet layer after clicking **OK**.
Layers	Specifies which design layers will be visible in the viewport
Display Planar Objects	Select to display 2D planar objects, when the viewport’s view is other than Top/Plan
Project Screen Objects	Select to display 2D objects associated with the screen plane, when the viewport’s view is other than Top/Plan
Classes	Specifies which classes will be visible in the viewport
Scale	Specifies the viewport scale relative to the page; select a scale or choose Custom and enter the scale value in **Custom Scale**
Custom Scale	When a custom scale is selected, enter the scale value
View	Specifies the orientation of the design layers displayed in the viewport; select a cardinal view or choose Custom and then click **Set View** to specify the view
Set View	When a custom view is selected, click **Set View** to open the 3D Rotation dialog box for the entry of custom view parameters (see “Rotating Precisely” on page 1151 for more information)
Rendering	Specifies the render mode for the viewport; some modes enable the **Render Settings** button, to specify rendering parameters

Parameter	Description
Render Settings	Available when the selected **Rendering** mode requires parameters to be set. See the following: • Sketch settings: “Applying Sketch Styles to Viewports” on page 1502 • OpenGL settings: “OpenGL” on page 1590 • Custom Renderworks settings: “Custom Renderworks Options” on page 1600 • Artistic Renderworks settings: “Artistic Renderworks Options” on page 1601 • Hidden Line, Dashed Hidden Line, and Final Shaded Polygon settings: “Line Render Options” on page 1592
RW Background (Renderworks required)	Select a Renderworks background from either the default content or the current file’s content to use as a background for the viewport; see “Resource Libraries” on page 219
Projection	Select the projection type for the viewport (see “Projection” on page 1142)
Perspective Type	For Perspective projection, select the type of perspective, or choose Custom and specify the perspective distance
Perspective Dist	For custom perspectives, enter the perspective distance

3. Enter the desired parameters and click **OK**.
4. If a sheet layer does not already exist in the file, the New Sheet Layer dialog box opens automatically to create one. Click **OK**.

   The viewport is created on the designated sheet layer, and the sheet layer becomes active.

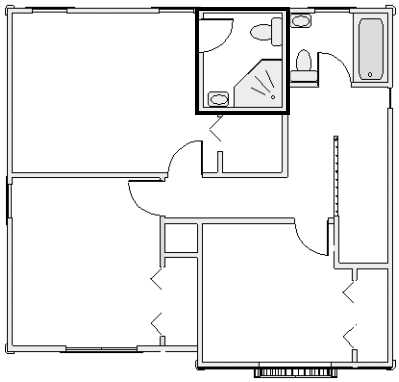
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Sheet Layer Viewport by Cropping
Linking the Camera View to a Sheet Layer Viewport
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

Creating a Sheet Layer Viewport by Cropping

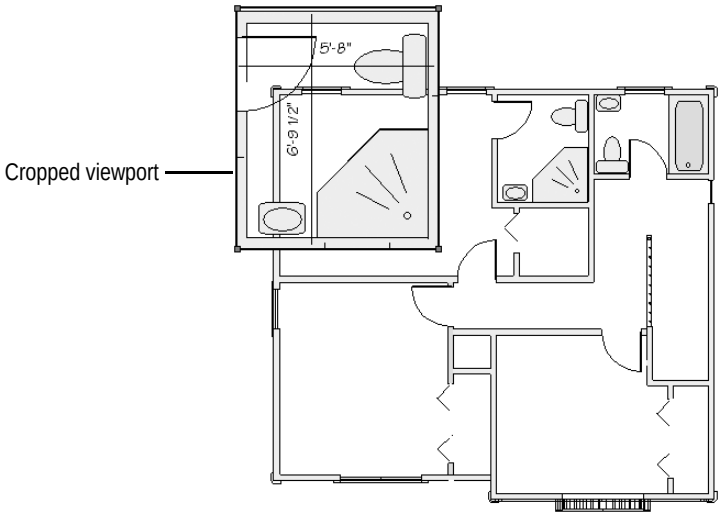
To create a cropped viewport either from a design layer or from an existing uncropped viewport on a sheet layer:

1. Make active the existing design layer or sheet layer that will display in the viewport.
2. Create a 2D object such as a rectangle, circle, or polyline. The 2D object must define an area; for example, a 2D line cannot be used. Crop objects are automatically placed in the screen plane (see “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152). Position the 2D object on the design layer or existing uncropped viewport to delimit the area to be included in the new viewport. The fill of a viewport cropping object is always None; however, the pen style can be set from the Attributes palette.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



3. If the cropped viewport is being created from a design layer, select the 2D object. If the cropped viewport is being created from a sheet layer, select both the 2D object and the uncropped viewport.
4. Select **View > Create Viewport**.
5. An alert dialog box asks whether the object should be used as the viewport's crop. Select **Yes** (click **Always do the selected action** to always use a selected 2D object as a crop object when creating viewports).
6. The Create Viewport dialog box opens. Enter a viewport name and drawing title, and select the sheet layer to place it on. The remaining viewport parameters are initially set to be the same as the design layer properties (for design layers) or selected viewport (for sheet layers). Change the parameters as needed.
7. Click **OK**.

   The viewport, cropped by the selected 2D object, is created on the specified sheet layer.
8. By default, the crop object is not visible. To change the visibility of the crop object, select the viewport and select the **Crop Visible** setting in the Object Info palette.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Sheet Layer Viewport from a Design Layer
Linking the Camera View to a Sheet Layer Viewport
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

D Creating Multiple Viewports Simultaneously

The **Create Multiple Viewports** command generates 2D drawings from a 3D model and creates up to seven sheet layer viewports configured with several orthographic views and one isometric view of the model.

If you run the command from a sheet layer, the viewports are added to that sheet layer. If you run the command from a design layer, the viewports are added to a new sheet layer that is created automatically.

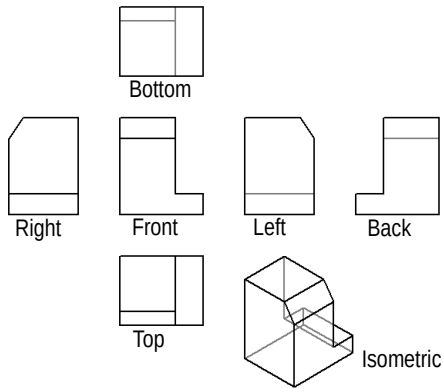
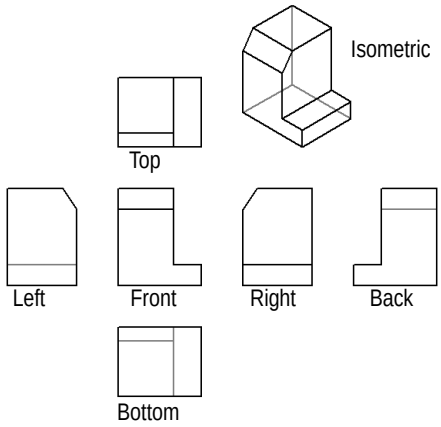
To create multiple viewports simultaneously:

1. Select **View > Create Multiple Viewports**.

   The Create Multiple Viewports dialog box opens.

2. Specify the desired viewport scale, views, and angle projection method, and then click **OK**.

Click to show/hide the parameters.

Parameter	Description
Viewport Scale	Select the viewport scale relative to the page
Method	
First Angle Projection	Create views using the first angle projection method; by default, the front, top, and left views are selected when the drawing units are metric 
Third Angle Projection	Create views using the third angle projection method; by default, the front, top, and right views are selected when the drawing units are imperial 

Parameter	Description
Views	
Front, Top, Right, Left, Bottom, Back, Isometric	Select the view(s) to create in the drawing

Viewports are created at the designated layer scale, using the current layer and class visibility and print area settings, with the rendered style set to hidden line rendering. Viewports are aligned horizontally and vertically, separated by a fixed distance, and centered on the sheet layer.

3. Optionally, configure the viewports' layer and class settings (Active, Show, or Gray Others only), annotate the viewport, or modify the rendering style or other viewport parameters.

   For more information, see "Setting Class and Design Layer Visibility for Viewports and Saved Views" on page 195, "Creating Annotations for Sheet Layer Viewports" on page 1653, and "Viewport Properties" on page 1636.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewports

Creating Design Layer Viewports

A sheet layer viewport displays a full or cropped view of one or more design layers, which you can change as needed and not affect the original drawing. For example, change the viewport's layer and class visibilities, use a different render mode, or add annotations and dimensions. (See "Creating Sheet Layer Viewports" on page 1616.)

Design layer viewports provide different functionality, for more flexibility. Like a sheet layer viewport, a design layer viewport can display design layers from the current file in a full or cropped view; unlike a sheet layer viewport, it can include one or more design layers that are referenced from another file.

Like a sheet layer viewport, in a design layer viewport you can control layer and class visibility, and create layer and class overrides. However, because it is an object on a design layer, a design layer viewport has the same view, scale, and render mode as everything else on the layer. You can use 2D and 3D drawing tools to add objects to the design layer, but you cannot add annotations to a design layer viewport.

Design layer viewports replace the layer link functionality present in the Vectorworks Fundamentals product. Current layer links can be easily converted to a design layer viewport with the **Modify > Convert > Convert to Viewport** command (Vectorworks Design Series required). (See "Converting Layer Links" on page 1859.)

The search criteria used in worksheets and in the Script Editor allow you to filter out items from design layer viewports, to prevent unwanted duplicates in schedules.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Design Layer Viewport from an Internal Design Layer
Creating a Design Layer Viewport by Cropping
Creating a Referenced Design Layer Viewport
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

D Creating a Design Layer Viewport from an Internal Design Layer

To create a design layer viewport that displays a design layer in the same file:

1. Select **View > Create Viewport**. Alternatively, from either the Organization dialog box or the Navigation palette, select the Viewports tab, and click **New**.
2. The Create Viewport dialog box opens. The scale, view, and render mode of the viewport are determined by the design layer on which it is placed; they cannot be changed here. Set the other viewport parameters as desired. After the viewport has been created, additional parameters become available; see “Editing a Design Layer Displayed in a Viewport” on page 1649.

Click to show/hide the parameters.

Parameter	Description
Viewport Name	Specifies the viewport name; this name must be unique in the document
Drawing Title	Specifies a descriptive name for the viewport. The drawing title name is limited to 63 characters.
Create on Layer	Select the design layer in the current file where the viewport will be created, or select New Design Layer to create a layer
Source	Displays the name of the file that contains the design layers to display in the viewport. If Current Document is not displayed here, click **Select Source** to open the Select Viewport Source dialog box; select **Current Document** as the source, and click **OK** to return to this dialog box. See “Creating a Referenced Design Layer Viewport” on page 1622 for information about referencing an external file.
Layers	Specifies which design layers will be visible in the viewport; see “Changing the Layer Properties of Sheet Layer or Design Layer Viewports” on page 1659
Display Planar Objects	Select to display 2D planar objects, when the viewport’s view is other than Top/Plan
Project Screen Objects	Select to display 2D objects associated with the screen plane, when the viewport’s view is other than Top/Plan
Classes	Specifies which classes from the source file will be visible in the viewport; see “Changing the Layer Properties of Sheet Layer or Design Layer Viewports” on page 1659
Remaining fields	The remaining fields are not available for design layer viewports

3. Click **OK**.

   The viewport is created on the designated design layer, and the design layer becomes active. The viewport can be cropped, as described in “Cropping Sheet Layer or Design Layer Viewports” on page 1652.

Creating Design Layer Viewports
Creating a Design Layer Viewport by Cropping
Creating a Referenced Design Layer Viewport
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

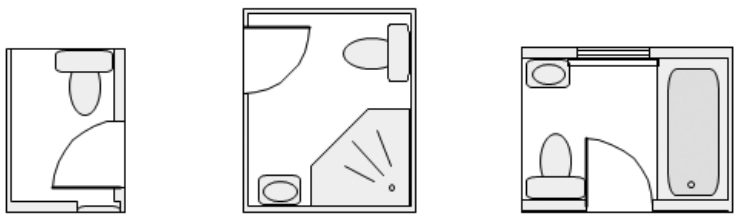
Creating a Design Layer Viewport by Cropping

The viewport and the design layer that displays in it must both be in the same file.

To create a design layer viewport by cropping:

1. Access the design layer that will display in the viewport.
2. Create a 2D object such as a rectangle, circle, or polyline. The 2D object must define an area; for example, a 2D line cannot be used. Crop objects are automatically placed in the screen plane (see “Planar Modes of 2D Objects: Screen Plane and Layer Plane” on page 152). Position the 2D object on the design layer to delimit the area to be included in the new viewport. The fill of a viewport crop object is always None; however, the pen style can be set from the Attributes palette.
3. Select the 2D crop object, and then select **View > Create Viewport**.
4. An alert dialog box asks whether the object should be used as the viewport’s crop. Select **Yes** (also select **Always do the selected action** to always use a selected 2D object as a crop object when creating viewports).
5. The Create Viewport dialog box opens. Select the design layer on which to create the viewport. The **Source** must be the current document. Specify the design layers and classes to display in the viewport. (See “Changing the Layer Properties of Sheet Layer or Design Layer Viewports” on page 1659 and “Changing the Class Properties of Sheet Layer or Design Layer Viewports” on page 1661.)
6. Click **OK**.

   The viewport, cropped by the selected 2D object, is created on the specified design layer.
7. By default, the crop object is not visible. To change the visibility of the crop object, select the viewport and select the **Crop Visible** setting in the Object Info palette.



1st Floor Powder Room | 2nd Floor Master Bath | 2nd Floor Hall Bath

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Design Layer Viewports
Creating a Design Layer Viewport from an Internal Design Layer
Creating a Referenced Design Layer Viewport
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

Creating a Referenced Design Layer Viewport

Referencing allows you to use information from other Vectorworks files in your file, including design layers, classes, and resources (such as hatches, worksheets, or symbols). There are two ways to reference design layers that are in other Vectorworks files:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- The default method in the Vectorworks Design Series products is to create a design layer viewport and then reference the desired design layers from the master file into the viewport, as described here. One advantage to this method is that all of the layers, classes, and resources from the master file are not automatically imported into the target file.
- In the Vectorworks Fundamentals product, design layers are imported into the target file when they are referenced. For backward compatibility, the Vectorworks Design Series products support this method. See “Adding and Editing Layer Import References” on page 210.

A reference for a design layer viewport can be created when the viewport is created, or it can be created ahead of time from the References tab of the Organization dialog box.

To create a design layer viewport that references a design layer in another file:

1. If the current file uses layer import referencing, switch to design layer viewport referencing. (From the References tab of the Organization dialog box, click **Settings**, and click the Design layer viewports option on the Reference Settings dialog box. Any existing referenced layers are automatically converted into referenced design layer viewports.)
2. Select **View > Create Viewport**. Alternatively, from either the Organization dialog box or the Navigation palette, select the Viewports tab, and click **New**.
3. The Create Viewport dialog box opens. Enter a **Viewport Name** and **Drawing Title**, and then select the design layer on which to create the viewport.
4. Click **Select Source** to open the Select Viewport Source dialog box, and enter information about the referenced document.

Click to show/hide the parameters.

Parameter	Description
Current Document / External Document	Select **External Document** to indicate that the design layers that will display in this viewport are in a different file
Existing Reference	If the master file is already a reference in this file, select this option, and then select the reference from the list
New Reference	Select this option if this is a new reference, and click **Choose** to open the Open Current-Version Drawing dialog box; locate the master file and click **Open** to return to this dialog box. The path to the selected file displays below the **New Reference** field. Referenced files must be the same Vectorworks software version as the target file. Circular references are not allowed; for example, if file A references file B, and file B references file C, then file C cannot reference file A. The exception is if one of the references uses the referencing in the Vectorworks Fundamentals product (layer import).
Save reference location as	Maintains either an absolute or relative file path reference from the current file to the referenced file. Use the absolute path when the location of the referenced file with respect to the current file is not going to change. Use the relative path when the files might be moved to another computer or platform; as long as the relative path between the files remains the same, the reference can be found. Both files must be saved on the save volume to select this option. The **Source File** path displays either an absolute or relative path, depending on the selection.

Parameter	Description
Automatically update out of date reference during file open	Updates the reference each time the target file is opened; when deselected, the reference is updated only when **Update** is clicked from the References tab of the Organization dialog box
Save referenced cache to disk	Saves a copy of the referenced data with the target file. When this option is deselected, a copy of the referenced data is not saved, which means that the target file size is smaller; the referenced data is refreshed when the target file is opened.

5. Click **OK** in the Select Viewport Source dialog box to return to the Create Viewport dialog box.
6. Specify the design layers and classes to display in the viewport (see “Changing the Layer Properties of Sheet Layer or Design Layer Viewports” on page 1659 and “Changing the Class Properties of Sheet Layer or Design Layer Viewports” on page 1661).
7. Click **OK**.

   The viewport is created on the designated design layer, and the design layer becomes active. The viewport can be cropped, as described in “Cropping Sheet Layer or Design Layer Viewports” on page 1652.

   The master file displays on the References tab of the Organization dialog box; for details about how to edit, update, or delete references, see “Workgroups and Referencing” on page 207.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Design Layer Viewports
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

D Creating Section Viewports

A section viewport creates a cross section view of a model, but leaves the model intact. The section viewport can display a 2D cross section view of only the objects that intersect the section line, or, additionally, the 3D geometry that remains on the indicated side of an infinite plane passing through the section line.

Create a live section view from one or more design layers, from a clip cube on a design layer, or from a viewport on a sheet layer or design layer. By creating several section viewports, models can be analyzed and presented effectively. The section views can be updated as the model changes, and their attributes and appearance can be easily changed.

A section viewport can be created on either a sheet layer or a design layer; the functionality and purpose of the two types of section viewports are different.

Section viewports on sheet layers can include annotations and automatic drawing coordination of the sheet and drawing numbers; also, if there are multiple viewports on a sheet layer, each can a have different view and scale. These types of sections are always Top/Plan since they are on sheet layers, and are created for construction drawings. To get a 3D “look,” switch to Perspective **Projection**.

Section viewports on design layers are useful when, for example, a detail section is needed in a Front rendered view. A design layer section viewport allows comparison between a section and an elevation view. In a team environment, the design layer section viewport can be referenced in other files.

> For sheet layer section viewports, to automatically coordinate the sheet numbers and drawing numbers among the sheet borders, drawing labels, and section markers in a file, select **Use Automatic Drawing Coordination** in the Display tab of the Document Preferences. This feature keeps references up to date, even when drawings are edited or moved to a different layer.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

D Creating a Section Viewport on a Sheet Layer

When you create a section viewport, you can place it on either a sheet layer or design layer. Section viewports on sheet layers can have annotations and automatic drawing coordination of the sheet and drawing numbers; also, if there are multiple viewports on a sheet layer, each can a have different view and scale.

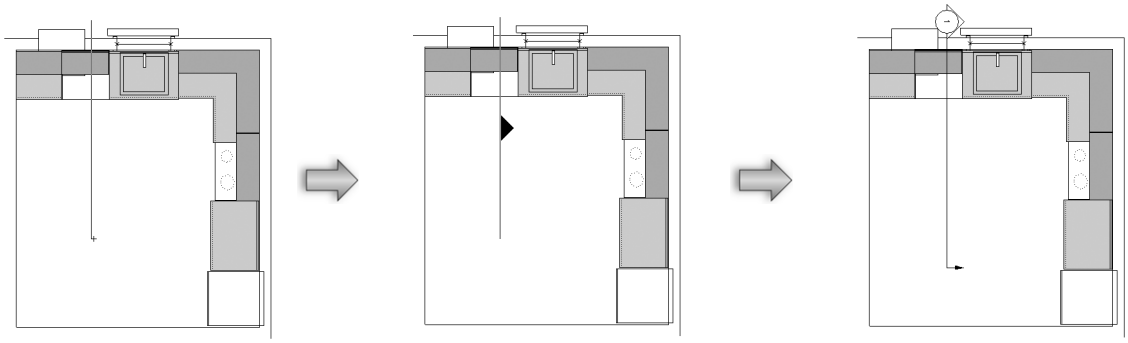
> If you need to draw a detailed section, or to reference the section viewport into other files, create the viewport on a design layer instead. (See “Creating a Section Viewport on a Design Layer” on page 1628.)

Create a section view from a design layer, from a clip cube, or from a viewport on a sheet layer. A section viewport that was created from a design layer or clip cube can be updated when changes are made to the design layer. However, a section viewport that was created from a viewport does not maintain a connection to the viewport that created it. It updates when the design layers that are visible in the source viewport change.

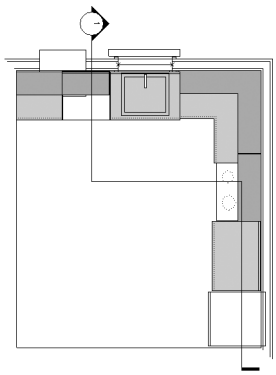
To create a section viewport on a sheet layer:

1. Prepare to create the viewport as follows:
   - To create a section view from an active design layer, set the layer to Top/Plan view by selecting **View > Standard Views > Top/Plan**.
   - To create a section view from an existing viewport, select a non-sectioned viewport object. The viewport object must be in Top, Bottom, Left, Right, Front, or Back view orientation.
   - To create a section view from an existing clip cube object, use the **Selection** tool to highlight the vertical face of the clip cube where the section will begin. (See “Viewing a Model with the Clip Cube” on page 1155.) Right-click (Windows) or Ctrl-click (Mac) to open the context menu.
2. Select **View > Create Section Viewport** (for a design layer or viewport), or select **Create Section Viewport** from the context menu (for a clip cube).
3. Draw the section line to create the cutting plane on the design layer or viewport. For a clip cube, skip to step 4; the section line is created automatically from the clip cube face that was used to create the section viewport.

   Click in the drawing and drag the mouse to begin drawing the marker line. Click to mark the end of the line, and then click to indicate the side of the line to look toward (keep), which is indicated by a black arrow. Double-click to end the line.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



To create a broken section line, click in the drawing and draw the first segment. Indicate which side of the drawing to show in the viewport. Click and drag to draw additional segments; broken section line segments are always parallel or perpendicular to each other. Double-click to end the broken line.



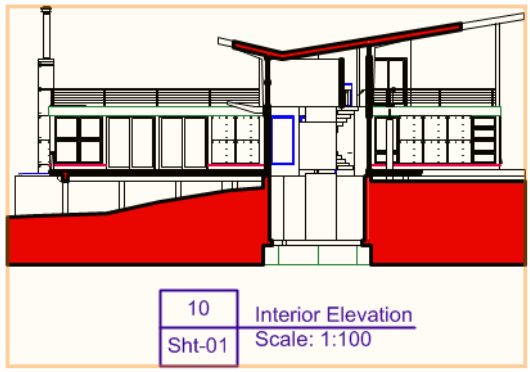
4. The Create Section Viewport dialog box opens, set to the parameters of the active design layer or non-section viewport. Change the parameters as needed.

Click to show/hide the parameters.

Parameter	Description
Name Viewport as Dwg No./Sheet No.	Select this option to automatically set the **Viewport Name** to be a combination of the **Drawing Number** and the **Sheet Number** assigned to this viewport. Deselect this option to enter a custom **Viewport Name**.
Viewport Name	If **Name Viewport as Dwg No./Sheet No.** is not selected, enter a descriptive name for the viewport; this name must be unique in the document
Create on Layer	Select the sheet layer where the viewport will be created, or select New Sheet Layer to create a sheet layer. If there are no sheet layers present and a new one is not created now, you will be prompted to create a sheet layer after clicking **OK**.
Create Drawing Label	Select this option to create a drawing label in the annotation space of the section viewport
Drawing Number	The next sequential drawing number available on the selected **Sheet Layer** defaults automatically. This number displays on the section marker associated with this viewport, as well as on any drawing label in the viewport's annotations. This number must be unique on this sheet layer.

Parameter	Description
Drawing Title	Specifies a descriptive title for the section viewport. This name displays as the drawing title for any annotation objects (drawing labels, section markers, and sheet borders) that are added to the viewport. The drawing title name is limited to 63 characters. If **Use Automatic Drawing Coordination** is selected in document preferences, a change to this field for the viewport automatically changes the field for the viewport's drawing label, and vice versa.
Layers	Specifies which design layers are visible in the viewport
Display Objects beyond Section Plane	When selected, shows objects beyond the section plane; visible planar objects are sectioned if positioned at the section line
Display Planar Objects	Select to display 2D planar objects; visible planar objects are sectioned if positioned at the section line
Display Flattened	This option is available for viewports on design layers only
Classes	Specifies which classes are visible in the viewport
Scale	Specifies the viewport scale relative to the page; select a scale, or choose Custom and enter a **Custom Scale** value
Custom Scale	When a custom scale is selected, enter the scale value
Rendering	Specifies the render mode for the viewport; some modes enable the **Render Settings** button, to specify rendering parameters After the viewport is created, foreground rendering can be added for a composite rendering effect; see "Viewport Properties" on page 1636.
Render Settings	Available when the selected **Rendering** mode requires parameters to be set. See the following: • Sketch settings: "Applying Sketch Styles to Viewports" on page 1502 • OpenGL settings: "OpenGL" on page 1590 • Custom Renderworks settings: "Custom Renderworks Options" on page 1600 • Artistic Renderworks settings: "Artistic Renderworks Options" on page 1601 • Hidden Line, Dashed Hidden Line, and Final Shaded Polygon settings: "Line Render Options" on page 1592
RW Background (Renderworks required)	Select a Renderworks background from either the default content or the current file's content to use as a background for the viewport; see "Applying Renderworks Backgrounds" on page 1545
Projection	Select the projection type for the viewport (see "Projection" on page 1142)
Perspective Type	For Perspective projection, select the type of perspective, or choose Custom and specify the perspective distance
Perspective Dist	For custom perspectives, enter the perspective distance
Advanced Section Properties	Specifies advanced section viewport parameters defining the extent and attributes of the section view; see "Advanced Section Viewport Properties" on page 1645
Section Line Settings	Specifies the graphic attributes of the section line object; see "Section Lines and Section-Elevation Markers" on page 1630

5. Click **OK**. A section line object is created in the design layer, or is added to the annotations of the existing non-sectioned viewport. A section viewport is created on the selected sheet layer, and the drawing switches to that sheet layer, displaying the new section viewport.



   By default, the cross section areas (along the plane where the section was cut) are displayed in red. These areas are in the Section Style class, if you want to change the fill color.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Section Viewports
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

D Creating a Section Viewport on a Design Layer

When you create a section viewport, you can place it on either a sheet layer or design layer. Section viewports on design layers are useful when a detailed section is needed, or when the section viewport needs to be referenced into other files.

> If you want to use annotations and automatic drawing coordination of the sheet and drawing numbers, or if you need multiple viewports on a sheet layer with different views and scales, create the viewport on a sheet layer instead. (See “Creating a Section Viewport on a Sheet Layer” on page 1625.)

Create a section view from a design layer, from a clip cube, or from a viewport on a sheet layer. A section viewport that was created from a design layer or clip cube can be updated when changes are made to the design layer. However, a section viewport that was created from a viewport does not maintain a connection to the viewport that created it. It updates when the design layers that are visible in the source viewport change.

To create a section viewport on a design layer:

1. Prepare to create the viewport as follows:
   - To create a section view from an active design layer, set the layer to Top/Plan view by selecting **View > Standard Views > Top/Plan**.
   - To create a section view from an existing viewport, select a non-sectioned viewport object. The viewport object must be in Top, Bottom, Left, Right, Front, or Back view orientation.
   - To create a section view from an existing clip cube object, use the **Selection** tool to highlight the vertical face of the clip cube where the section will begin. (See “Viewing a Model with the Clip Cube” on page 1155.) Right-click (Windows) or Ctrl-click (Mac) to open the context menu.
2. Select **View > Create Section Viewport** (for a design layer or viewport), or select **Create Section Viewport** from the context menu (for a clip cube).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. Draw the section line to create the cutting plane on the design layer or viewport. For a clip cube, skip to step 4; the section line is created automatically from the clip cube face that was used to create the section viewport.

   Click in the drawing and drag the mouse to begin drawing the marker line. Click to mark the end of the line, and then click to indicate the side of the line to look toward (keep), which is indicated by a black arrow. Double-click to end the line.

   To create a broken section line, click in the drawing and draw the first segment. Indicate which side of the drawing to show in the viewport. Click and drag to draw additional segments; broken section line segments are always parallel or perpendicular to each other. Double-click to end the broken line.

4. The Create Section Viewport dialog box opens. After you select the design layer on which to create the viewport, several parameters that apply only to section viewports on sheet layers become unavailable; they are not included in the parameters table. Enter the desired parameters.

   The scale of a design layer section viewport is the same as the layer where it is placed. The rendering mode of the current layer is also used to render the design layer section viewport.

Click to show/hide the parameters.

Parameter	Description
Viewport Name	If **Name Viewport as Dwg No./Sheet No.** is not selected, enter a descriptive name for the viewport; this name must be unique in the document
Create on Layer	Select the design layer where the viewport will be created, or select New Design Layer to create a design layer. If there are no design layers present and a new one is not created now, you will be prompted to create a design layer after clicking **OK**.
Layers	Specifies which design layers are visible in the viewport
Display Objects beyond Section Plane	When selected, shows objects beyond the section plane, on the indicated side of the section; deselect to show only the objects that intersect the section line
Display Planar Objects	Select to display 2D planar objects; visible planar objects are sectioned if positioned at the section line
Display Flattened	This option becomes available after you select a design layer in the **Create on Layer** field. When selected, the viewport always displays a "flattened" section view, similar to a section viewport on a sheet layer; deselect this option to have the viewport display the current view selected for the design layer.
Classes	Specifies which classes are visible in the viewport
Advanced Section Properties	Specifies advanced section viewport parameters defining the extent and attributes of the section view; see "Advanced Section Viewport Properties" on page 1645
Section Line Settings	Specifies the graphic attributes of the section line object; also sets the default attributes for the section lines for future section viewports you create (see "Section Lines and Section-Elevation Markers" on page 1630)

5. Click **OK**. A section line object is created in the design layer, or is added to the annotations of the existing non-sectioned viewport. A section viewport is created on the selected design layer, and the drawing switches to that design layer, displaying the new section viewport.

   Initially, the view is set to Top/Plan, but this can be changed, and the section can be displayed in any view (the **Flyover** tool can also be used to view the section). Additionally, there is an option to display a flattened version of the section, which can be used to create section drawings or details.

   A design layer section viewport can be cropped, but it does not contain an annotation space.

Creating Section Viewports
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

D Section Lines and Section-Elevation Markers

Section lines and section-elevation markers look very similar on a drawing, but they are created differently.

A section line is automatically created when a section viewport is created. The section line graphically indicates the cutting plane of the section viewport and shows the orientation of the section view. Specify the graphic properties of the section line (such as the marker style), when you create the section viewport. To access a section line from its section viewport, click **Section Line Instances** from the Object Info palette (see "Section Line Instances" on page 1631).

> A section line that has been pasted from a copy, duplicated, or mirrored from a section line that was associated with a section viewport becomes an "unlinked" section line. It displays with black and yellow stripes and the **Section Viewport** field in the Object Info palette displays "Not Linked."

A section-elevation marker is graphically similar to a section line, but it is not linked to any viewport. Use the **Section-Elevation Marker** tool from the Dims/Notes tool set to insert a marker, as a reference line graphic for sections and elevations, or as a cutting plane graphic. Select the Preferences option from the Tool bar to specify the graphic properties of the object before you create it.

After creation, edit both section lines and section-elevation markers as follows:

- Use the Object Info palette to edit the graphic properties (such as the marker style).
- Use the Attributes palette to apply attributes (such as the fill or pen color).
- Use options on the **Text** menu to control the text appearance (such as the font or size), or assign a text style.
- Use the **Selection** or **Reshape** tool to edit the objects (see "Modifying Section Lines Graphically" on page 1656).

Creating Section-Elevation Markers

 To insert a section-elevation marker:

1. Select the **Section-Elevation Marker** tool from the Dims/Notes tool set.
2. Select the Preferences option from the Tool bar, and set the default parameters for section-elevation markers.
3. Click to place one end of the section-elevation marker.
4. To insert a single-segment marker, drag to determine the marker length.

   To insert a multi-segment marker, click to define each segment. Because a multi-segment marker is a polyline, the methods of drawing and editing polylines apply (see "Creating Polylines" on page 298).
5. Double-click to finish placing the section-elevation marker.

Creating Section Viewports
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

Section Line Instances

Section line instances associated with a section view can be added to or deleted from design layers or viewport annotations. In addition, the Section Line Instances dialog box provides an easy way to navigate from a section viewport to its associated section lines.

To edit section line instances or navigate to a section line:

1. Select the section viewport whose section line you want to navigate to or edit.
2. Click **Section Line Instances** from the Object Info palette.

   The Section Line Instances dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Viewport	Displays the name of the selected section viewport
Design Layers tab/ Viewports tab	Displays the names of all design layers in the file (on the Design Layers tab) and all sheet layer viewports in the file (on the Viewports tab). Each design layer or viewport that contains a section line associated with the selected section viewport has a check mark to the left of its name.
Activate	When a design layer or viewport with a check mark is selected, navigates to the section line instance

3. To add another instance of the section line to a different design layer or to the annotation for a different viewport, click in the column next to that design layer or viewport, and then click **OK**. Alternatively, to remove a section line instance from a layer or viewport, click the checked column (which removes the check mark), and then click **OK**.

   Deleting all section line instances does not delete the section viewport, and new instances can be created at any time. However, deleting a section viewport deletes all section line instances.

4. The section line instances can also be used as a navigation tool to access a particular section line. Select the section line and click **Activate** to switch to the design layer or viewport; the section line is selected for any modifications.

   To return from a section line instance to the associated section viewport, click **Navigate to Section Viewport** from the Object Info palette of a selected section line.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Section Viewports
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

Creating Section Viewports from Unlinked Section Lines

An unlinked (orphan) section line is disconnected from its associated section viewport, possibly because the section line was pasted from a copy, duplicated, or mirrored. It displays as a black and yellow line, and “Not Linked” is displayed in **Section Viewport** on the Shape tab of the Object Info palette.

Section viewports can be created from unlinked section lines located on a design layer, sheet layer, or while in edit annotation mode.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To create a section viewport from an unlinked section line:

1. Select the unlinked section lines. If on a sheet layer, include the viewport to be sectioned in the selection set.

   Each selected section line creates a new section viewport.

2. Select **View > Create Section Viewport**.

   The Create Section Viewport dialog box opens (see "Creating Section Viewports" on page 1624). If multiple unlinked section lines were selected, the parameters specified apply to all section viewports created.

3. Click **OK** to create a section viewport for each selected section line.

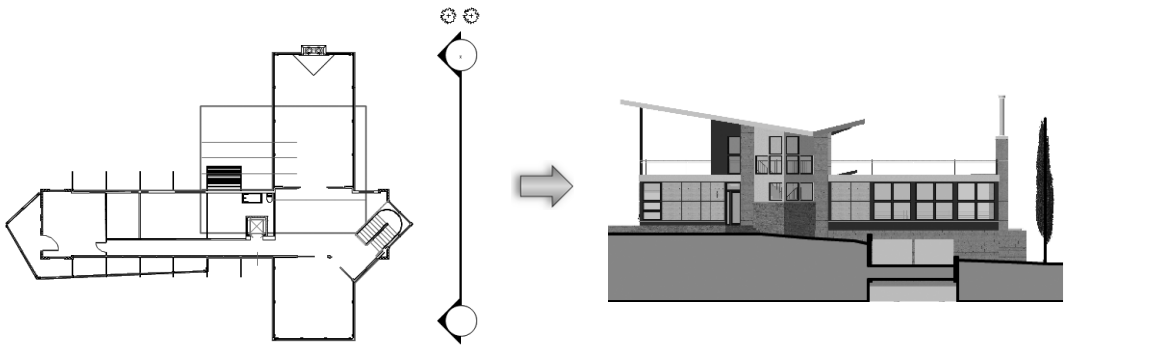
Creating Section Viewports
Section Lines and Section-Elevation Markers
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

D Creating and Sectioning Elevation Views

Section viewports are useful for creating images that show multiple design layers from a certain point of view.

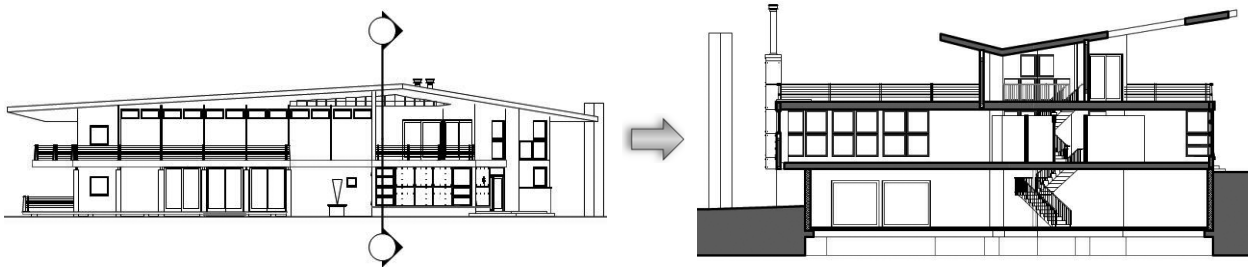
For example, to show an exterior elevation of a building model, draw a section line outside of the model on one of the design layers and make all of the necessary layers visible in the section viewport. To show an interior view of the same model, do the same thing, but draw the section line through the model at the appropriate location.

Alternatively, if you already have a (non-sectioned) sheet layer viewport that shows multiple design layers in an elevation view, you can draw the section line through the viewport at the appropriate location.



In Top/Plan view on one of the design layers, draw a section line looking toward the model

Make all of the design layers visible in the section viewport to create an exterior elevation view



On a viewport that shows an elevation view, draw a section line to create an interior elevation view

Creating Section Viewports

Creating Detail Viewports

The **Create Detail Viewport** command creates a cropped sheet layer viewport that shows a detail view of any part of a drawing. A detail viewport can be created from a design layer, another sheet layer viewport, or a section viewport.

The crop object used to create the detail viewport becomes a callout object that is linked to the detail viewport for easy navigation between layers. If the detail callout object is moved or reshaped, the detail viewport is updated accordingly. Additionally, the detail callout is automatically labeled with the drawing number and sheet number of the detail viewport. If the detail viewport is renumbered or moved to another layer, the callout marker is updated automatically.

To create a detail viewport:

1. Create a crop object on a design layer (in Top/Plan view), on a sheet layer viewport (sectioned or not), or inside a viewport annotation group.
2. Select the crop object and then select **View > Create Detail Viewport**.

   The Create Detail Viewport dialog box opens. Different options are available, depending on where the crop object is located (design layer, viewport, or section viewport). Specify the desired parameters.

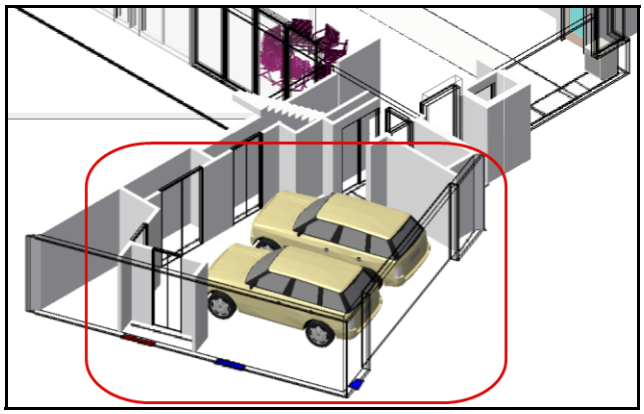
Click to show/hide the parameters.

Parameter	Description
Name Viewport as Dwg No./Sheet No.	Select this option to automatically set the **Viewport Name** to be a combination of the **Drawing Number** and the sheet layer name. Deselect this option to enter a custom **Viewport Name**.
Viewport Name	If **Name Viewport as Dwg No./Sheet No.** is not selected, enter a descriptive name for the viewport. This name displays for the viewport on the Navigation palette and Organization dialog box. The name must be unique in the document.
Create on Layer	Select the sheet layer where the viewport will be created, or select New Sheet Layer to create a sheet layer. If there are no sheet layers present and a new one is not created now, you will be prompted to create a sheet layer after clicking **OK**.
Create Drawing Label	Select this option to create a drawing label in the annotation space of the detail viewport
Drawing Number	The next sequential drawing number available on the selected sheet layer defaults automatically. This number displays on the drawing label associated with this viewport, as well as on the detail callout. The number must be unique on this sheet layer.
Drawing Title	Specifies a descriptive title for the viewport; this name displays on the drawing label associated with this viewport. The drawing title name is limited to 63 characters.
Layers	Specifies which design layers are visible in the viewport
Display Planar Objects	Select to display 2D planar objects
Project Screen Objects	Available if the detail viewport is being created from a design layer or a non-section viewport; when selected, displays 2D objects associated with the screen plane
Display Objects Beyond Section Plane	Available if the detail viewport is being created from a section viewport; when selected, shows objects beyond the section plane

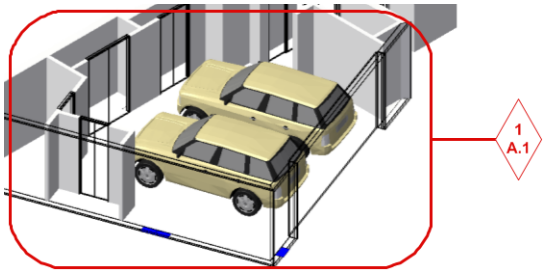
Parameter	Description
Classes	Specifies which classes are visible in the viewport
Scale	Specifies the viewport scale relative to the page; select a scale, or choose Custom and enter a **Custom Scale** value
Custom Scale 1:	When a custom scale is selected, enter the scale value
Rendering	Available if the detail viewport is being created from a viewport; some modes enable the **Render Settings** button, to specify rendering parameters
Render Settings	Available if the detail viewport is being created from a viewport, and the selected **Rendering** mode requires parameters to be set. See the following: • Sketch settings: “Applying Sketch Styles to Viewports” on page 1502 • OpenGL settings: “OpenGL” on page 1590 • Custom Renderworks settings: “Custom Renderworks Options” on page 1600 • Artistic Renderworks settings: “Artistic Renderworks Options” on page 1601 • Hidden Line, Dashed Hidden Line, and Final Shaded Polygon settings: “Line Render Options” on page 1592
RW Background (Renderworks required)	Available if the detail viewport is being created from a viewport and one of the Renderworks **Rendering** modes is selected. Select a Renderworks background from either the default content or the current file’s content to use as a background for the viewport; see “Resource Libraries” on page 219.
Detail Callout Settings	Opens a dialog box to set options for the detail callout object; see “Properties of Detail Callouts” on page 1642

3. Click **OK** to close the Create Detail Viewport dialog box. The specified sheet layer is activated, and the detail viewport is placed in the center of the sheet.

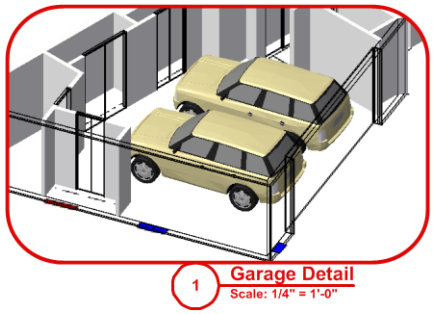
   Additionally, the original crop object is converted into a detail callout object. If the detail viewport was created from a design layer, the detail callout is created on the design layer. If the detail viewport was created from a viewport, the detail callout is created in the annotation space of that viewport. The callout includes the drawing number and sheet number of the detail viewport associated with it.



Draw a crop object over a portion of the viewport and select View > Create Detail Viewport



A detail callout object is created in the original viewport's annotations



A detail viewport is created on the specified sheet layer; the detail viewport and detail callout are linked

4. You can create additional instances of the callout on other design layers or in the annotations of other viewports, if needed. See "Detail Callout Instances" on page 1635.
5. To delete a detail viewport and its associated callout, delete the viewport.

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports
Detail Callout Instances

Detail Callout Instances

Detail callout instances associated with a detail viewport can be added to or deleted from design layers or viewport annotations. In addition, the Detail Callout Instances dialog box provides an easy way to navigate from a detail viewport to its associated detail callout.

To edit detail callout instances or navigate to a detail callout:

1. Select the detail viewport whose callout you want to navigate to or edit.
2. Click **Detail Callout Instances** from the Object Info palette.

   The Detail Callout Instances dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Viewport	Displays the name of the selected detail viewport
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Design Layers tab/ Viewports tab	Displays the names of all design layers in the file (on the Design Layers tab) and all other sheet layer and section viewports in the file (on the Viewports tab). Each design layer or viewport that contains a detail callout associated with the selected detail viewport has a check mark to the left of its name.
Activate	When a design layer or viewport with a check mark is selected, navigates to the detail callout instance: • If the detail callout instance is on a design layer, activates that design layer and centers the detail callout object in the screen • If the detail callout instance is in a viewport, activates the sheet layer of the viewport and enters annotation editing mode for the viewport

3. To add another instance of the detail callout to a different design layer or to the annotations for a different viewport, click in the column next to that design layer or viewport, and then click **OK**. Alternatively, to remove a detail callout instance from a layer or viewport, click the checked column (which removes the check mark), and then click **OK**.

   Deleting all detail callout instances does not delete the detail viewport, and new instances can be created at any time. However, deleting a detail viewport deletes all detail callout instances.

4. The detail callout instances can also be used as a navigation tool to access a particular callout. Select the detail callout and click **Activate** to switch to the design layer or viewport; the detail callout is selected for any modifications.

   To return from a detail callout instance to the associated detail viewport, click **Navigate to Viewport** from the Object Info palette of a selected detail callout.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Detail Viewports
Viewport Properties
Modifying Viewports
Viewport Status
Updating Viewports

Viewport Properties

Once it has been created, edit the viewport in the Object Info palette, or select **Properties** from the viewport's context menu to open the Properties dialog box. Edit the parameters as needed.

A viewport is assigned to the None class when it is created, and its visibility is controlled by the class settings when that class is not active.

Click to show/hide the parameters.

Parameter	Description
Rotation	Sets the viewport rotation; if the viewport was created from a rotated plan view (Vectorworks Design Series required), this parameter can be used to reset the viewport to the world coordinate system
Lock Position (design layer viewport) (Vectorworks Design Series required)	Select to prevent the viewport from being moved accidentally
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Crop	Indicates whether the selected viewport has been cropped (see "Cropping Sheet Layer or Design Layer Viewports" on page 1652)
Crop Visible	If the viewport has been cropped, select to display the crop object
Update	Click to update the viewport to reflect any changes that have occurred since the viewport was created or last updated
Description (design layer viewport) (Vectorworks Design Series required)	Specifies a descriptive name for the viewport
Drawing Title	Specifies a descriptive title for the viewport. This name displays as the drawing title for any annotation objects (drawing labels, sheet borders, and section markers) that are added to the viewport. The drawing title name is limited to 63 characters. If **Use Automatic Drawing Coordination** is selected in document preferences, a change to this field for the viewport automatically changes the field for the viewport's drawing label, and vice versa. (Vectorworks Design Series required)
Drawing Number (sheet layer section viewport, detail viewport) (Vectorworks Design Series required)	The next sequential drawing number available on the selected **Sheet Layer** defaults automatically. This number displays on the section marker associated with this viewport, as well as on any drawing label in the viewport's annotations. This number must be unique on this sheet layer. Additionally, if **Text Auto-Fill** is selected for the detail callout, a change to this field for the viewport automatically changes the field for the detail callout.
Layers	Specifies which design layers are visible in the viewport and allows changes to some of the layer properties in the viewport; see "Changing the Layer Properties of Sheet Layer or Design Layer Viewports" on page 1659
Display Objects Beyond View Plane (section viewport, detail viewport) (Vectorworks Design Series required)	Shows or hides objects beyond the view plane
Display Planar Objects	Select to display 2D planar objects, when the viewport's view is other than Top/Plan. For section viewports, visible planar objects are sectioned if positioned at the section line (Vectorworks Design Series required).
Project Screen Objects	Select to display 2D objects associated with the screen plane, when the viewport's view is other than Top/Plan. Detail viewports must be created from a design layer or non-section viewport to enable this option.
Display Flattened (design layer section viewport) (Vectorworks Design Series required)	When selected, the viewport always displays a "flattened" section view, similar to a section viewport on a sheet layer; deselect this option to have the viewport display the current view selected for the design layer

Parameter	Description
Classes	Specifies which classes are visible in the viewport and allows changes to some of the class properties in the viewport, including changes to the properties for annotation or crop objects. Class visibilities can be overridden for a selected viewport; see “Changing the Class Properties of Sheet Layer or Design Layer Viewports” on page 1661.
Scale	Specifies the viewport scale relative to the page; select a scale, or choose Custom and enter a **Custom Scale** value
Custom Scale	When a custom scale is selected, enter the scale value
RW Camera (Renderworks required)	Indicates whether the viewport is linked to a Renderworks camera
View	Specifies the orientation of the design layers displayed in the viewport; select a cardinal view or choose Custom and then click **Set View** to specify the view The standard view of a selected viewport can be changed with the numeric keypad shortcut keys (see “Using Standard Views” on page 1141).
Set View	When a custom view is selected, click **Set View** to open the 3D Rotation dialog box for the entry of custom view parameters (see “Rotating Precisely” on page 1151 for more information)
Background/Foreground Render	Specifies the render mode(s) for the viewport. Select a background mode and specify the render settings, if any. For a composite effect, also select an optional foreground mode and specify any settings (Wireframe, Sketch, Hidden Line, or Dashed Hidden Line are the only render modes available for foreground rendering).
Background / Foreground Render Settings	Some render modes require parameters to be set; click the appropriate **Render Settings** button to specify them. See the following: • Wireframe settings: “Wireframe Options” on page 1589 • Sketch settings: “Applying Sketch Styles to Viewports” on page 1502 • OpenGL settings: “OpenGL” on page 1590 • Custom Renderworks settings: “Custom Renderworks Options” on page 1600 • Artistic Renderworks settings: “Artistic Renderworks Options” on page 1601 • Hidden Line, Dashed Hidden Line, and Final Shaded Polygon settings: “Line Render Options” on page 1592
RW Background (Renderworks required)	Select a Renderworks background from either the default content or the current file’s content to use as a background for the viewport; see “Applying Renderworks Backgrounds” on page 1545 If the Renderworks Background choices are not available, the background selection is controlled by the Renderworks style in effect. Set the background in the Renderworks style instead; see “Renderworks Styles” on page 1596.
Projection	Select the projection type for the viewport (see “Projection” on page 1142)
Perspective Type	For Perspective projection, select the type of perspective, or choose Custom and specify the perspective distance
Perspective Dist	For custom perspectives, enter the perspective distance

Parameter	Description
Lighting Options	Click to change the ambient light parameters described in "Setting Lighting Options" on page 1571. By default, a viewport's ambient light is set according to the ambient light settings of the first visible design layer in the viewport. If there are no visible layers, the ambient light is set to on, with a color of white and a brightness of 35% (similar to the default ambient lighting for a design layer). If the lighting options are controlled by a Renderworks style that is currently in effect (Renderworks required), the Edit Renderworks Style dialog box opens instead; see "Renderworks Styles" on page 1596.
Advanced Properties	Opens the Advanced Viewport Properties dialog box; see "Advanced Sheet Layer Viewport Properties" on page 1642, "Advanced Design Layer Viewport Properties" on page 1644, or "Advanced Section Viewport Properties" on page 1645
Source (design layer viewport) (Vectorworks Design Series required)	Indicates the source file that contains the original design layer(s) that display in the viewport
Reverse Direction (section viewport) (Vectorworks Design Series required)	Switches to view the other side of the section line
Section Line Instances (section viewport) (Vectorworks Design Series required)	Lists the section line instances present in the file (see "Section Line Instances" on page 1631)
Detail Callout Instances (detail viewport) (Vectorworks Design Series required)	Lists the detail callout instances present in the file (see "Detail Callout Instances" on page 1635)

Once a section viewport is created (Vectorworks Design Series required), you may need to add or remove items that display in the section view. To do so, first use the **Convert to Lines**, **Convert to Polygons**, or **Convert to Group** command to convert the viewport to another form. (This conversion results in a group of lines and other primitives; it is no longer a viewport and can't be updated to reflect drawing changes.) The conversion will produce different results, depending on which command is used and on how the viewport is rendered at the time of the conversion.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

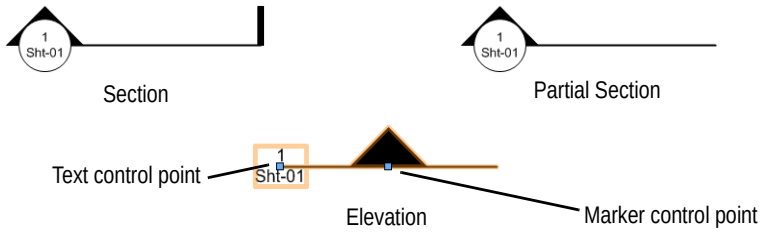
Properties of Section Lines and Section-Elevation Markers
Properties of Detail Callouts
Advanced Sheet Layer Viewport Properties
Advanced Design Layer Viewport Properties
Advanced Section Viewport Properties
Creating Sheet Layer Viewports
Creating Design Layer Viewports
Creating Section Viewports
Creating Detail Viewports
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

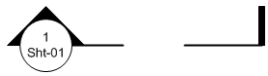
Modifying Viewports
Viewport Status
Updating Viewports

Properties of Section Lines and Section-Elevation Markers

Parameters display on the Object Info palette for selected section lines and section-elevation markers. Many of these parameters also display on the Section Line Settings dialog box, which you can access when you create a section viewport. Edit the parameters as needed.

Click to show/hide the parameters.

Parameter	Description
Section Viewport (Section Lines only)	Displays the name of the viewport associated with the section line; a section line that is not associated with a viewport is labeled “Not Linked”
Reverse Direction (Section Lines only)	Reverses the side to display when objects beyond the section plane are visible
Navigate to Section Viewport (Section Lines only)	Navigates to the section viewport associated with the section line
Rotation (Section-Elevation Markers only)	Sets the marker rotation; if the marker was created from a rotated plan view, this parameter can be used to reset the marker to the world coordinate system
Text Style	Select a text style from either the default content or the current file’s content. To use the style associated with the class, select <Class Text Style>. To format the text using options on the **Text** menu, select <Un-Styled>.
Configuration	Select the section line configuration • Section: draws a section marker at both ends of a straight or broken section line • Partial Section: draws a section marker at the beginning of a straight or broken section line • Elevation: draws a section marker along a straight section line at a control point, which can be dragged to a new location 
Flip (Section-Elevation Markers only)	Reverses the direction of the marker

Parameter	Description
Set Marker Style(s) (for Section-Elevation Markers) or Section Marker Style(s) (for Section Lines)	Opens the Set Marker Style dialog box, to select the section marker type at each end of the section line; markers can be the same (**Match Beginning**) or different (**Differentiate**) at each end of the section line. Select from either the default content or the current file's content; see "Resource Libraries" on page 219.
Marker Size	Specifies the size of the section marker(s), measured from the marker's insertion point to the top of its longest point. Changes to the marker size do not affect the text size.
Use Gapped Line (Section and Partial Section configurations only)	Displays the section line with a gapped line 
Gap Line Length	If **Use Gapped Line** is selected, specifies the length of the solid section line at each end of the section line
Text Auto-Rotate	When selected, the marker text always displays in the same orientation; when deselected, the text is rotated according to the **Text Rotation** value
Text Both Ends (Section configuration only)	Displays text at the end of the marker, as well as the beginning
Text Style	Specifies the display style for the **Drawing Number** and **Sheet Number** text; if Dwg. No.-Sheet No. or Sheet No.-Dwg. No. is selected, specify a **Text Separator** to place between the text items
Text Rotation	When **Text Auto-Rotate** is not selected, specifies the rotation of the marker text
Text Sheet %	Specifies the scale value for the **Sheet Number** portion of the marker text. Enter a number larger than 100 to make the text larger. Enter a number below 100 to make it smaller. The scale value must be larger than 1.
Text Separator	For the Dwg. No.-Sheet No. and Sheet No.-Dwg. No. text styles, specifies the separator to place between the **Drawing Number** and **Sheet Number** text
Text Auto-Fill (Section Lines only)	For section line objects associated with a section viewport, automatically displays the Drawing Number of the referenced section viewport, and the viewport's Sheet Number. Deselect this option to enter the **Drawing Number** and **Sheet Number** manually.
Drawing Number	Enter the **Drawing Number** of the referenced section viewport; this value defaults for section lines when **Text Auto-Fill** is selected
Sheet Number	Enter the **Sheet Number** of the referenced section viewport; this value defaults for section lines when **Text Auto-Fill** is selected

Viewport Properties
Creating Section Viewports

Properties of Detail Callouts

Parameters display for selected detail callouts on the Object Info palette. Many of these parameters also display on the Detail Callout Settings dialog box, which you can access when you create a detail viewport. Edit the parameters as needed. If there are multiple instances of the detail callout, all instances are updated.

Click to show/hide the parameters.

Parameter	Description
Detail Viewport	Displays the name of the viewport
Navigate to Viewport	Accesses the detail viewport associated with this detail callout
Detail Callout Graphic Options	Opens a dialog box to set options for the marker attached to the callout object. Select a **Marker Symbol** from the list of those available. The **Shoulder Length** is the distance from the shoulder point of the leader line to the center of the marker. **Leader Line** sets the style and thickness of the leader line. **Tag Position** sets the position of the marker relative to the callout bubble.
Marker Size	Specifies the size of the callout marker, measured from the marker's insertion point to the top of its longest point. Changes to the marker size do not affect the text size.
Text Style	Specifies the display style for the **Drawing Number** and **Sheet Number** text; if Dwg. No.-Sheet No. or Sheet No.-Dwg. No. is selected, specify a **Text Separator** to place between the text items
Text Rotation	Specifies the rotation of the marker text
Text Sheet	Specifies the scale value for the **Sheet Number** portion of the marker text. Enter a number larger than 100 to make the text larger. Enter a number below 100 to make it smaller. The scale value must be larger than 1.
Text Separator	For the Dwg. No.-Sheet No. and Sheet No.-Dwg. No. text styles, specifies the separator to place between the **Drawing Number** and **Sheet Number** text
Text Auto-Fill	Automatically displays the Drawing Number of the referenced detail viewport, and the detail viewport's Sheet Number. Deselect this option to enter the **Drawing Number** and **Sheet Number** manually.
Drawing Number	Enter the **Drawing Number** of the referenced detail viewport; this value defaults when **Text Auto-Fill** is selected
Sheet Number	Enter the **Sheet Number** of the referenced detail viewport; this value defaults when **Text Auto-Fill** is selected

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewport Properties
Creating Detail Viewports

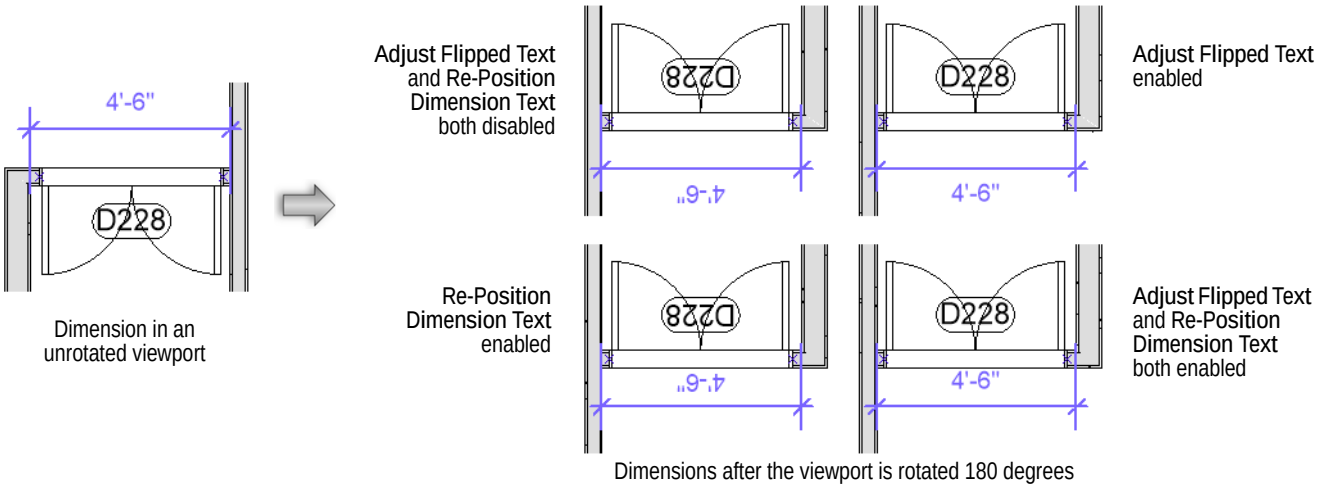
Advanced Sheet Layer Viewport Properties

To access additional sheet layer viewport parameters, click **Advanced Properties** from the Object Info palette or Properties dialog box of a selected viewport. The Advanced Viewport Properties dialog box opens. These settings affect the viewport display only; they do not change the original design layer(s). Edit the parameters as needed.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Line Weight Scale	Enter a value larger than 1.0 to increase the viewport line weights, or a value below 1.0 (but larger than 0) to decrease the line weights
Marker Scale	Enter a value larger than 1.0 to increase the viewport marker size, or a value below 1.0 (but larger than 0) to decrease the marker size
Line Type Scale	Enter a value larger than 1.0 to increase the length and spacing of viewport line type segments, or a value below 1.0 (but larger than 0) to decrease the length and spacing of line type segments
Hatch Line Scale	Enter a value larger than 1.0 to increase the spacing between viewport hatch lines, or a value below 1.0 (but larger than 0) to decrease the spacing between hatch lines
Text Scale	Enter a value larger than 1.0 to increase the text size in viewports, or a value below 1.0 (but larger than 0) to decrease the text size; only associated viewport text is affected. Graphic objects that are part of the text item, such as a reference marker container or callout bubble, are scaled with the text. Dimensional objects inside plug-in objects, such as dimensions within a bubble grid object, are not scaled. Plug-in objects, such as a North arrow object, are not scaled if they have been placed while editing the viewport in Edit Annotation mode.
Page Symbol Scaling	These settings affect page-based symbols in the viewport (see “Symbol Types” on page 237)
Symbol Scale	Sets a scale factor for page-based symbols; a scale factor of less than one decreases the size of the symbol relative to its definition at a 1:1 scale, while a factor of more than one increases its size
Attribute Scaling	These settings affect the attributes (such as line weight) of page-based symbols in the viewport
Use Symbol Factor	Uses the **Symbol Scale** factor to scale the attributes
Use Individual Factors	Uses the other, individual scale factors in the Advanced Properties dialog box, such as the **Line Weight Scale,** to scale the page-based symbol attributes
Show Wall Components	Displays or hides wall and slab components in Top/Plan view, regardless of the document preferences detail display setting (see “Document Display Preferences” on page 60)
Render Gray Layers Transparent	Design layers with a visibility set to “Gray” are rendered as transparent, similar to the transparent effect achieved with the **Unified View** command (see “Unified Layer View” on page 1152)
Black and white only	Changes all colors in the viewport to black or white; this is useful for displaying two viewport copies on the same sheet layer, with one in color and the other in black and white. However, if the document preferences display setting is black and white, viewports will also display as black and white.
Adjust Flipped Text	Re-orients rotated and flipped text in the viewport so that it is always readable. The text box stays in the rotated and flipped position relative to the dimension line, but the text itself is re-oriented. (This setting overrides the flipped text preference for the document; see “Document Display Preferences” on page 60.)

Parameter	Description
Re-Position Dimension Text	Re-positions rotated and flipped dimension text in the viewport according to the text position in the dimension standard. The text itself stays in the rotated and flipped orientation, but the text box position relative to the dimension line is changed. (This setting overrides the flipped text preference for the document; see “Document Display Preferences” on page 60.)
Preview	Displays the viewport with a preview of the advanced settings



Viewport Properties
Creating Sheet Layer Viewports

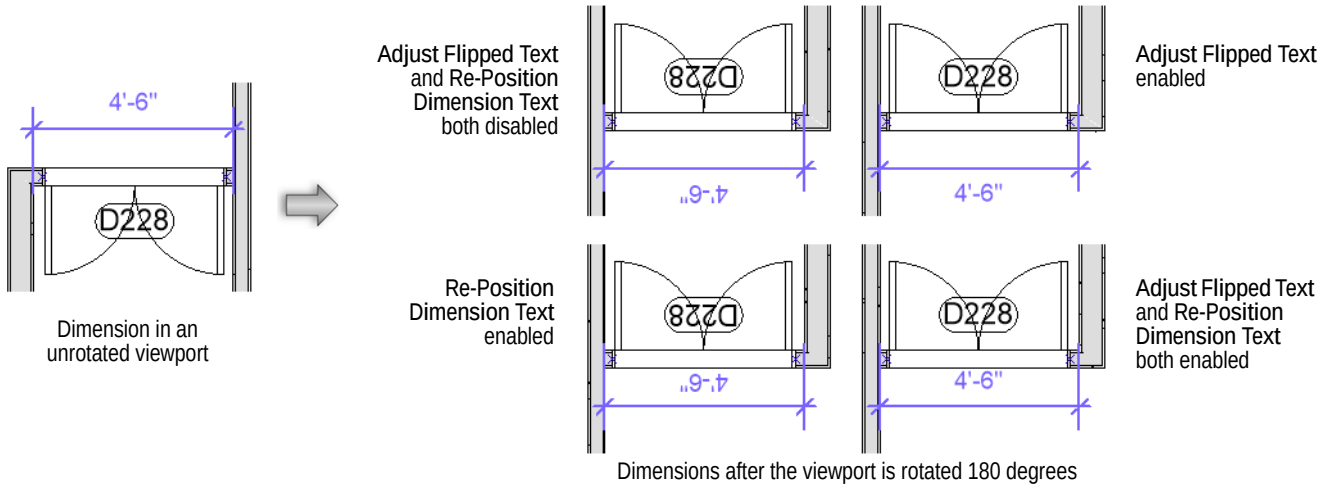
Advanced Design Layer Viewport Properties

To access additional design layer viewport parameters, click **Advanced Properties** from the Object Info palette of a selected viewport. The Advanced Viewport Properties dialog box opens. These settings affect the viewport display only; they do not affect the original design layer(s). Edit the parameters as needed.

Click to show/hide the parameters.

Parameter	Description
Show Wall Components	Displays or hides wall components in Top/Plan view, regardless of the document preferences detail display setting (see “Document Display Preferences” on page 60)
Black and white only	Changes all colors in the viewport to black or white; this is useful for displaying two viewport copies on the same design layer, with one in color and the other in black and white. However, if the document preferences display setting is black and white, viewports will also display as black and white.
Adjust Flipped Text	Re-orients rotated and flipped text in the viewport so that it is always readable. The text box stays in the rotated and flipped position relative to the dimension line, but the text itself is re-oriented. (This setting overrides the flipped text preference for the document; see “Document Display Preferences” on page 60.)

Parameter	Description
Re-Position Dimension Text	Re-positions rotated and flipped dimension text in the viewport according to the text position in the dimension standard. The text itself stays in the rotated and flipped orientation, but the text box position relative to the dimension line is changed. (This setting overrides the flipped text preference for the document; see “Document Display Preferences” on page 60.)
Preview	Displays the viewport with a preview of the advanced settings



Viewport Properties
Creating Design Layer Viewports

Advanced Section Viewport Properties

Advanced properties define the extent and attributes of the sheet layer or design layer section viewport. Specify them when the viewport is created, or after the viewport has been created; edit the properties from either the Object Info palette or the Properties dialog box.

To specify the advanced properties of a section viewport:

1. From the Create Section Viewport dialog box, click **Advanced Section Properties**.

   Alternatively, select the viewport, and then from the Object Info palette or the Properties dialog box, click **Advanced Properties**.

   The Advanced Section Properties dialog box opens.

2. Click the Extent tab to specify either an infinite section view, or the length, depth, and height of a finite section viewport.

   Finite sections are useful when you want to create interior elevations. For example, draw a section line across a particular room of a building, and set the layer visibility only for the room’s floor to create an interior elevation of only that room. For an even more precise interior elevation, use a clip cube to create the section viewport; the extents are automatically set according to the clip cube’s shape.

Click to show/hide the parameters.

Parameter	Description
Length Range	Specifies the length range of the section viewport
Infinite	Cuts the section by an infinite plane
Limited by Section Line Length	Limits the section only to the length defined by the section line
Depth Range	Specifies the depth range of the section viewport
Infinite	All objects on the indicated side of the section line are displayed
Finite	Displays objects on the indicated side of the section line up to a specified depth (the depth can be indicated graphically by dragging a dotted line on the design layer; see “Modifying Section Lines Graphically” on page 1656)
Height Range	Specifies the height range of the section viewport
Infinite	Does not limit the section to a Z range
Finite	Limits the display of the section viewport to a range defined by the Start and End Height Z values

3. Click the Attributes tab to specify the appearance of objects on and beyond the section plane. Objects on the section plane can maintain an individual profile, or can be divided into structural and non-structural groups to display them differently according to class settings.

Click to show/hide the parameters.

Parameter	Description
Section Plane	Objects on the section plane can be merged into a single profile, maintain individual profiles, or be divided into groups of structural and non-structural elements (useful for displaying cabinets and walls differently, for example)
Merged Cross Sections	Merges the sectioned objects’ cross section profiles into one profile
Create Structural and Nonstructural Groups	Merges the structural objects’ cross section profiles into one group, and the non-structural objects’ cross section profile into another, allowing them to have a different class setting
Separate Cross Sections	Generates a separate cross section for each sectioned object
Use Attributes of Original Objects	Each sectioned object is displayed with the attributes of its original object
Attribute Class / Structural Class	Specifies the line and fill class for the section profiles; by default, the Section Style class is applied to the cross sections, or you can select New to create a new class. If **Create Structural and Nonstructural Groups** is selected, the class selected here applies only to structural elements.
Nonstructural Class	When **Create Structural and Nonstructural Groups** is selected, specifies the line and fill class for non-structural elements, or you can select New to create a new class
Objects Beyond Section Plane	Specifies the fill and line style for objects beyond the section plane (for section viewports that display objects beyond the section plane)
Fill	
Use Original	Uses the fill style of the original objects for objects beyond the section plane

Parameter	Description
Use Class	Sets the fill style of the objects beyond the section plane by class. Select a class or select New to create a new class.
Line Style	Changes section viewport settings for line thickness and line type
Use Original	Uses the line thickness and line type of the original objects for the objects beyond the section plane
Use Class	Sets the line thickness and line type of the objects beyond the section plane by class. Select a class or select New to create a new class.
Scale Factors	Changes section viewport display scale settings for lines and line styles
Line Weight Scale	Enter a number larger than 1.0 to make the section line weight thicker. Enter a number below 1.0 to make the line weight thinner. The scale value must be above zero.
Line Type Scale	A line type could be used for a section viewport if the section is in a class with a line type (in the **Attribute Class**/**Structural Class** parameter) or the objects beyond the section plane have been set to a class with a line type (in the **Use Class** parameter). Enter a number larger than 1.0 to make the line weight of the line type sections thicker. Enter a number below 1.0 to make them thinner. The scale value must be above zero.

4. Click the Display tab to specify the section viewport display properties.

Click to show/hide the parameters.

Parameter	Description
Cast Shadows of Objects Removed by Section	When rendering, includes shadows cast by objects that are not included in the section viewport, for a more realistic effect (especially for interior elevations) This parameter is not available for design layer section viewports.
Show Wall and Slab Components	Displays the wall and slab components in 3D, regardless of the document preferences detail display setting (see "Vectorworks Display Preferences" on page 50), or component class visibility settings. The top and bottom wall lines of components do not display in section viewports, so that stacked walls display without a line break.
Merge Adjacent Components with Same Fill	If adjacent wall and/or slab components have the same fill, merges the fills so that they display as a single unit
Black and White Only	Changes all colors in the section viewport to black or white; this is useful for displaying two section viewport copies on the same sheet layer, with one in color and the other in black and white. However, if the document preferences display setting is black and white, viewports will also display as black and white.
Adjust Flipped Text	Re-orients rotated and flipped text in the viewport so that it is always readable (regardless of the flipped text Vectorworks preference; see "Document Display Preferences" on page 60)
3D Conversion Resolution	Sets the segmentation resolution for curved surfaces in a live section, regardless of the Vectorworks preferences resolution setting (see "3D Preferences" on page 54). Changing the resolution of an existing section viewport causes the viewport to become out of date; the higher resolution settings require longer update times.

5. Click **OK** to close the dialog box and save the changes.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewport Properties
Creating Section Viewports

Modifying Viewports

There are several ways to modify viewports; their appearance can be completely different from the original design layers, for presentation purposes.

- Modify the settings for the viewport in the Object Info palette.
- Modify the viewport with various 2D and 3D tools and commands.
- Crop the viewport.
- Edit the design layer(s) that display in the viewport.
- Add annotations and dimensions to the viewport.
- Edit or delete a linked Renderworks camera (Renderworks required).
- Change the properties of the viewport's layers and classes.

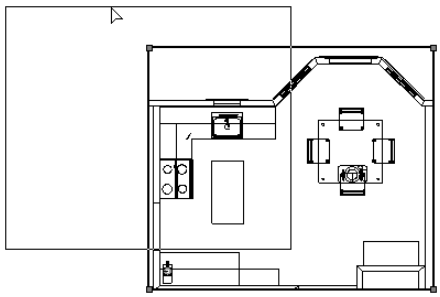
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Moving and Editing Viewports
Editing a Design Layer Displayed in a Viewport
Cropping Sheet Layer or Design Layer Viewports
Creating Annotations for Sheet Layer Viewports
Editing a Linked Renderworks Camera
Modifying Section Viewports and Section Lines
Editing Detail Viewport Drawing Labels
Editing Detail Callout Objects
Changing the Layer Properties of Sheet Layer or Design Layer Viewports
Changing the Class Properties of Sheet Layer or Design Layer Viewports
Viewport Properties
Viewport Status
Updating Viewports

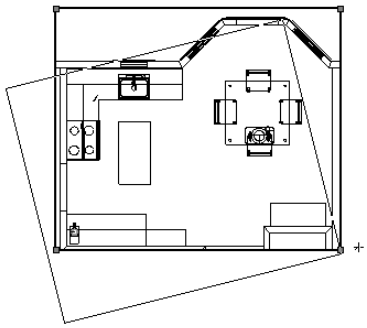
Moving and Editing Viewports

A sheet layer viewport can be edited like most 2D objects. For information on editing tools and commands, see "Editing Objects" on page 997. 3D tools cannot be used on sheet layers. However, a 3D object can be copied from a design layer and pasted on a sheet layer. A design layer viewport (Vectorworks Design Series required) cannot be pasted on a sheet layer.

- Use the **Cut**, **Copy**, and **Paste** commands to copy or paste a viewport on its original sheet layer or another sheet or design layer. Use the **Selection** tool to drag a viewport to a new position (or edit the X- and Y-axis positions in the Object Info palette). Press the Delete key to delete a selected viewport.



- Use the **Move** and **Rotate** commands and the **Rotate** and **Mirror** tools to move, rotate, or mirror a viewport. The viewport can be split by the **Split** tool (in Line Split mode), and clipped with the **Clip** tool.



- Use the **Scale Objects** command to scale a viewport. Any crop objects in the viewport are also scaled, as are annotations and dimensions. Viewport text, however, is not scaled unless **Scale Text** is selected in the Scale Objects dialog box.
- To scale a viewport with the mouse, select the viewport and use the **Modify > Group** command to turn it into a group object. Click and drag a handle to resize the viewport as needed. Then use the **Modify > Ungroup** command to change it back into a viewport object.
- Use the **Modify > Lock** and **Modify > Unlock** commands to lock and unlock viewports.
- Use the **Eyedropper** tool to transfer attributes from one viewport to another; see “Transferring Attributes” on page 1095.
- Use 2D drawing tools on sheet layers to create borders, title blocks, and so on.
- A sheet layer viewport can be copied and pasted into an image-editing application. The dpi setting of the sheet layer affects the resolution of the pasted image. Depending on the platform and the image-editing application, the resolution of the pasted image may still not be optimal; in this case, the **File > Export Image File** command offers control over the exported area, dimensions, resolution, and file type.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports

Editing a Design Layer Displayed in a Viewport

To edit a design layer that is displayed in a viewport:

1. Select the viewport.
2. Select **Modify > Edit Viewport**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a viewport, and select **Edit** from the context menu.

   The Edit Viewport dialog box opens. The options available depend on whether the viewport is on a sheet layer or design layer (Vectorworks Design Series required).

Click to show/hide the parameters.

Parameter	Description
Annotations (sheet layer viewport)	Creates or edits viewport annotations and dimensions
Crop	Creates or edits a cropped viewport, and optionally displays the viewport outside of the crop area when in Edit Crop mode
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Display Viewport Outside Crop	Displays the viewport outside of the crop with wireframe rendering; objects outside of the crop can be snapped to when creating or editing a crop shape. This option is only available when the Crop edit mode is selected.
Gray Outside Crop	If **Display Viewport Outside Crop** is selected, displays the area outside of the crop in gray. This option is only available when the Crop edit mode is selected.
Design Layer	Navigates to the selected design layer to edit objects contained in the viewport
Display using Viewport Attributes	Changes the file's view parameters and layer and class visibilities to match those of the viewport. If the **Navigate Back to Viewport** option is also selected, the file's layer and class visibilities return to their original status when you return to the viewport; otherwise, the file's attributes remain the same as the viewport's. This option is only available when the Design Layer edit mode is selected.
Add Reference Crop Object	When a viewport has been cropped, this option displays the crop on the design layer so that edits can be made to the design layer while knowing the position of the crop object. This option is only available when the Design Layer edit mode is selected.
Navigate Back to Viewport	Temporarily adds a **Return to Viewport** button to the design layer that returns you to the viewport when the edit to the design layer is complete (similar to an Edit Group operation). This option is only available when the Design Layer edit mode is selected. To exit to the design layer instead of exiting to the viewport from design layer editing mode, press Shift + Esc. Alternatively, right-click (Windows) or Ctrl-click (Mac) in the drawing area, and select **Exit Viewport** from the context menu.
Display with Clip Cube (Vectorworks Design Series required)	If you are editing a section viewport, and **Navigate Back to Viewport** is selected, this option displays the design layer with a clip cube whose dimensions match the length, depth, and height ranges set for the viewport. Portions of the design layer outside the clip cube are not visible during editing. You can adjust the faces and orientation of the clip cube as needed, and then either create a new section viewport, or update the current viewport. See "Viewing a Model with the Clip Cube" on page 1155 for details. This option is only available when the Design Layer edit mode is selected.
Camera (Renderworks required)	Edits a Renderworks camera that is linked to the viewport view. The camera view can be changed or the camera can be deleted. See "Editing a Linked Renderworks Camera" on page 1654. If no Renderworks camera is linked to the viewport, select a camera to be linked. Alternatively, the view can be manipulated with the standard view tools (such as the Flyover tool, zoom level, and **View** menu commands), changing the viewport view upon exit.

Parameter	Description
Display Viewport Cache	If the viewport is currently in a render mode other than Wireframe, select **Display Viewport Cache** to display a cache image of the rendered viewport during editing; deselect to display a Wireframe view of the viewport. This option is only available when the Annotations or Crop edit mode is selected.
Keep the Edit View on Exit	When you return to the viewport after an edit, this setting maintains any view changes (zoom and view location) made during edits to the viewport annotation or the crop object. Deselect this option to return to the original viewport view settings after editing. This option is only available when the Annotations or Crop edit mode is selected.
Double Click	Sets the future behavior when a viewport is double-clicked, eliminating the display of this dialog box if desired. If the Edits the Design Layer option is selected, a double-click activates the design layer of the double-clicked object. If the object does not belong to a design layer, the Edit Viewport dialog box opens to select a design layer to edit. If Displays this Dialog is not selected, you can still open the dialog box: select **Modify > Edit Viewport**, or right-click (Windows) or Ctrl-click (Mac) the viewport and select **Edit** from the context menu.

3. Click **Design Layer** and select the design layer to edit from the list.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a viewport and select **Edit Design Layer** from the context menu to activate the design layer of the right-clicked object (if the right-clicked object does not belong to a design layer, the Edit Viewport dialog box opens).

4. Select the editing options:
   - Select **Display using Viewport Attributes** to view the design layer with the viewport attributes (orientation, projection, render mode, and layer and class visibilities). A rendered viewport displays the original design layer with the viewport's render mode; however, the design layer's render mode options for that mode are used.
   - Select **Add Reference Crop Object** to view the crop object on the design layer during editing. However, because the crop object is added to the design layer, it could become visible in other viewports that reference that area of the design layer.
   - Select **Navigate Back to Viewport** to easily return to the viewport when you are finished with the design layer edits. A colored border around the drawing window indicates that you are in an editing mode. The **Return to Viewport** button is visible in the top right corner of the drawing window.
   - If this is a section viewport, select **Display with Clip Cube** to use a clip cube while editing the design layer.
5. Click **OK** to activate the selected design layer.
6. If you selected **Navigate Back to Viewport**, a colored border displays around the drawing window. Do one of the following when your edits are complete:
   - Click **Return to Viewport** to return to the viewport. This saves changes to drawing objects, but it does not save clip cube edits.
   - Press Shift + Esc to exit to the design layer instead of exiting to the viewport. Alternatively, right-click (Windows) or Ctrl-click (Mac) in the drawing area, and select **Exit Viewport** from the context menu.
   - If the viewport has a clip cube, right-click (Windows) or Ctrl-click (Mac) the vertical cube face where the section line is located; select **Update Section Viewport** to save the clip cube edits. Then either click **Return to Viewport**, or select **Exit Viewport** from the context menu to exit editing mode.

- If the viewport has a clip cube, right-click (Windows) or Ctrl-click (Mac) any vertical cube face other than where the section line is located; select **Create Section Viewport** to create a new section viewport with its section line located at that cube face.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports

Cropping Sheet Layer or Design Layer Viewports

To crop a viewport:

1. Select the viewport.
2. Select **Modify > Edit Viewport**. The Edit Viewport dialog box opens (see “Editing a Design Layer Displayed in a Viewport” on page 1649 for a description of the dialog box parameters).
3. Click **Crop**.

   Choose whether to display the viewport outside of the crop area. Select **Display Viewport Outside Crop** to view the rest of the viewport; select **Gray Outside Crop** to view the area outside of the crop in gray. These options make drawing and editing easier, since objects outside the crop can be snapped to.

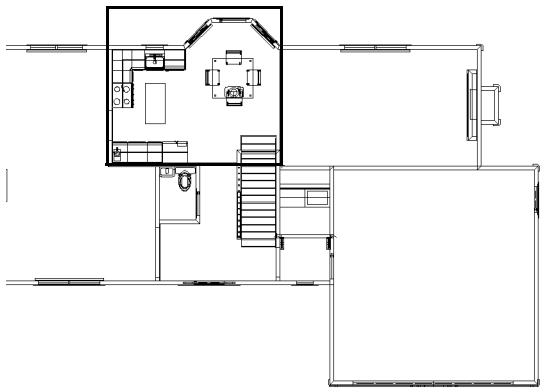
   To view other layer objects while in Edit Crop mode, select **Show other objects while in editing modes** on the Display tab of the Vectorworks preferences (see “Vectorworks Display Preferences” on page 50).
4. Click **OK** to enter Edit Crop mode.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a viewport and select **Edit Crop** from the context menu.

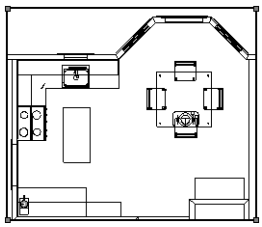
   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Viewport Crop** command becomes available from the **Modify** menu, and the **Exit Viewport Crop** button is visible in the top right corner of the drawing window.
5. Create a 2D object such as a rectangle, circle, or polyline. The 2D object must define an area; for example, a 2D line cannot be used. Position the 2D object to delimit the new viewport display area. The fill of a viewport cropping object is always None; however, the pen style can be set from the Attributes palette while in Edit Crop mode. Move and resize the 2D object as needed.

   Use the **Flyover** tool to adjust the view as necessary (see “Flyover” on page 1144).

   The bounding box of the crop object is also the perspective clip rectangle, if the viewport is in Perspective projection. Reshaping the crop object changes the perspective clip rectangle as well.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



6. Click **Exit Viewport Crop** to return to the sheet layer or design layer.



7. The cropped viewport displays; in the Object Info palette, the **Crop** status changes to Yes.
8. To change, replace, or delete the crop object, select the viewport and then select **Modify > Edit Viewport** to re-enter Edit Crop mode. Alternatively, right-click (Windows) or Ctrl-click (Mac) and select **Edit** from the context menu.

   To change the visibility of the crop object, change the **Crop Visible** setting in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports

Creating Annotations for Sheet Layer Viewports

Use the Edit Annotation mode to add annotations and dimensions in viewports, and to edit those annotations and dimensions later on.

To add annotations, including dimensions, to a viewport:

1. With the viewport's sheet layer active, select the viewport by clicking on it with the **Selection** tool.
2. Select **Modify > Edit Viewport**. The Edit Viewport dialog box opens (see "Cropping Sheet Layer or Design Layer Viewports" on page 1652 for a description of the dialog box parameters).
3. Click **Annotations** and then click **OK** to enter Edit Annotation mode.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a viewport and select **Edit Annotations** from the context menu.

   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Viewport** command becomes available from the **Modify** menu, and the **Exit Viewport Annotation** button is visible in the top right corner of the drawing window.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

4. Use the various dimension tools from the Dims/Notes tool set to add dimensions to the viewport (see “Dimensions” on page 1187). The dimension tools snap to the objects in the viewport as if you were dimensioning the design layer. The dimensions are automatically updated if the design layer object changes.

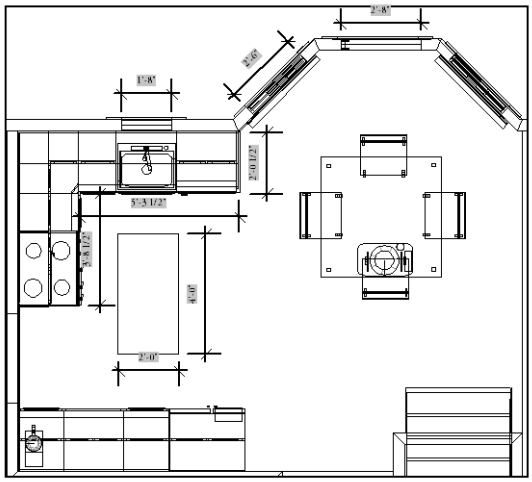
   Annotations are 2D objects that are placed on the screen plane. Therefore, a 2D object in the viewport must be dimensioned in Top/Plan view. A 3D object can be dimensioned in any view, but you must align the face that is to be dimensioned with the screen plane to get an accurate measurement.

   To view other objects on the sheet layer while in Edit Annotation mode, select **Show other objects while in editing modes** on the Display tab of the Vectorworks preferences (see “Vectorworks Display Preferences” on page 50).

   Text, callouts, and other annotations, as well as 2D objects, can be added to the viewport. The Vectorworks Design Series products contain additional annotation objects.

   The stacking order of selected annotations can be changed with the **Modify > Send** commands. To add graphical annotations to a viewport rendered with Hidden Line, use the inner or outer boundary mode of the **2D Polygon** tool (see “2D Polygon Tool” on page 303).

   Annotations are in viewport scale, not sheet layer scale.



5. Click **Exit Viewport Annotation** to exit Edit Annotation mode and return to the sheet layer.
6. To change, replace, or delete the viewport annotations, select the viewport and then select **Modify > Edit Viewport** to re-enter Edit Annotation mode.

   To change the visibility of the crop object, change the **Crop Visible** setting in the Object Info palette.

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports

Editing a Linked Renderworks Camera

You can edit a linked Renderworks camera, to change the associated sheet layer viewport’s view.

To edit a linked Renderworks camera:

1. Select the viewport.

2. Select **Modify > Edit Viewport**. The Edit Viewport dialog box opens (see "Editing a Design Layer Displayed in a Viewport" on page 1649 for a description of the dialog box parameters).
3. Click **Camera**.
4. An alert dialog box opens. Click **OK** to enter Edit Renderworks Camera mode.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a viewport and select **Edit Camera** from the context menu.

   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Renderworks Camera** command becomes available from the **Modify** menu, and the **Return to Viewport** button is visible in the top right corner of the drawing window.
5. The design layer that was active when the viewport was created is active, and the linked Renderworks camera object is selected. Edit the camera view as described in "Adjusting the Camera View" on page 1159.

   The camera can be deleted. The view and projection parameters are controlled by the viewport if the camera is deleted.

   If no Renderworks camera is linked to the viewport, select a camera to be linked. Alternatively, the view can be manipulated with the standard view tools (such as the **Flyover** tool, zoom level, and **View** menu commands), changing the viewport view upon exit.
6. Click **Return to Viewport** to return to the viewport once the Renderworks camera has been edited or deleted. The viewport's view, projection, and perspective distance are updated.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports

Modifying Section Viewports and Section Lines

Section viewports can be modified, cropped, annotated, and updated, the same as regular sheet layer viewports. See "Modifying Viewports" on page 1648 and "Updating Viewports" on page 1665.

The appearance of a section viewport can be completely customized, from the items it displays to the attributes of those items. Copies of a section viewport on a sheet layer can look completely different. Changes to a section viewport's appearance can be made by several methods; after changes are made, update the section viewport by clicking **Update** in the Object Info palette.

Modification	Method	Description
Change the section view	Click **Advanced Properties** from the Object Info palette of a selected section viewport, and modify the section view from the Extent tab	"Advanced Section Viewport Properties" on page 1645
Change the view attributes	Click **Advanced Properties** from the Object Info palette of a selected section viewport, and modify the attributes of the view from the Attributes tab	"Advanced Section Viewport Properties" on page 1645
Change the view direction of the section line	Click **Reverse Direction** from the Object Info palette of a selected section viewport	"Viewport Properties" on page 1636
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modification	Method	Description
Change the cross section appearance for sectioned items	The cross section appearance is set by the Section Style class. Edit the class to change the appearance of sectioned items. If the Section Style class is made invisible, the cross sections are not displayed.	"Setting Class Properties" on page 179
Override the layer settings from the design layer	Click **Layers** from the Object Info palette of a selected section viewport, and override the layer properties. Unlike a regular sheet layer viewport, the design layer stacking order in the section viewport cannot be changed.	"Changing the Layer Properties of Sheet Layer or Design Layer Viewports" on page 1659
Override the class settings from the design layer	Click **Classes** from the Object Info palette of a selected section viewport, and override the class properties of "by class" objects	"Changing the Class Properties of Sheet Layer or Design Layer Viewports" on page 1661
Change the location of the section line	Change the location of the section line with the **Selection** tool, and update the section viewport	"Modifying Section Lines Graphically" on page 1656
Create section viewports from unlinked section lines	Select the unlinked section lines and then select the **Create Section Viewport** command to create section viewports from the section lines	"Creating Section Viewports from Unlinked Section Lines" on page 1631
Add additional section line instances to design layers or viewports	Click **Section Line Instances** from the Object Info palette of a selected section viewport, and specify the design layers or viewports where section line instances should display	"Section Line Instances" on page 1631
Change the section line length, position, or type	Change the section line with the **Selection** tool or the **Reshape** tool, and update the section viewport	"Modifying Section Lines Graphically" on page 1656
Change the depth of a finite section	The depth can be changed graphically or by modifying the **Depth Range** in the Advanced Properties of the section viewport	"Modifying Section Lines Graphically" on page 1656, or "Advanced Section Viewport Properties" on page 1645

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Section Lines Graphically
Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports

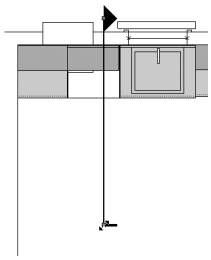
Modifying Section Lines Graphically

In addition to modifying section line parameters as described in "Section Lines and Section-Elevation Markers" on page 1630, the section line can be modified by changing its location, length, or shape.

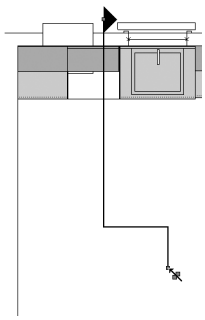
To modify a section line:

1. Navigate to the section line by clicking **Section Line Instances** from the Object Info palette of a selected section viewport.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

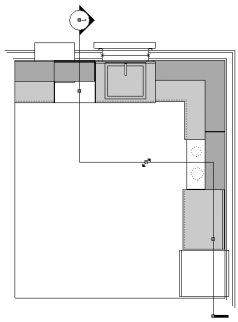
2. The section line is automatically selected for modification.
   - Move the section line to a new location with the **Selection** tool
   - Shorten, lengthen, or rotate the line by dragging an end point with the **Selection** tool
   - Add vertices and change a straight section line to a broken section line with the **Reshape** tool
   - Reshape the section line with the **Reshape** tool



Changing the length

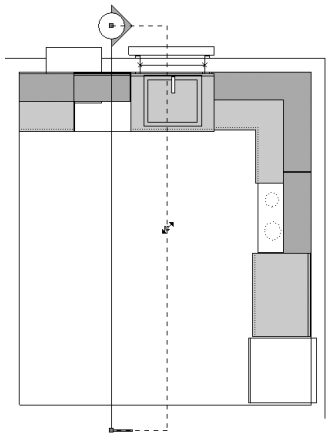


Adding vertices



Adjusting the position of vertices

3. A section view with a finite depth has a special control point on a dashed line. To adjust the depth, drag the control point with the **Selection** tool.



4. Return to the section viewport by clicking **Activate Section Viewport** from the Object Info palette. Click **Update** from the Object Info palette to reflect the section line changes in the section view.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports
Reshaping Objects

D Editing Detail Viewport Drawing Labels

After creation, the detail viewport drawing label may be edited, either to change its appearance, or to change the information that it displays. Edit the drawing label from within the detail viewport's annotation space.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

If **Use Automatic Drawing Coordination** is selected in document preferences, a change to the **Drawing Number** field for the detail viewport automatically changes the field for the viewport's drawing label, and vice versa. Additionally, if **Text Auto-Fill** is selected for the detail callout, a change to the **Drawing Number** field for the viewport automatically changes the field for the detail callout.

There are several ways to change the appearance of a detail viewport drawing label.

- Use the Attributes palette to change the label's line color or thickness
- Use the Object Info palette to change the object properties
- Use the **Text** menu to change the attributes of the label text (or apply a text style to it)
- Use the **Selection** tool to adjust the label position

To edit a detail viewport drawing label:

1. Select the detail viewport and then select **Modify > Edit Viewport**. The Edit Viewport dialog box opens.
2. Click **Annotations** and then click **OK** to enter Edit Annotation mode.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a viewport and select **Edit Annotations** from the context menu.

   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Viewport** command becomes available from the **Modify** menu, and the **Exit Viewport Annotation** button is visible in the top right corner of the drawing window.
3. Select the drawing label and edit it as needed. The parameters in the Object Info palette are described in "Creating Drawing Labels" on page 1244.
4. Click **Exit Viewport Annotation** to exit Edit Annotation mode and return to the sheet layer.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports
Reshaping Objects

Editing Detail Callout Objects

After creation, the detail callout may be edited, either to change the appearance of the callout itself, or to change the view in the detail viewport that is linked to the detail callout. The edit method depends on where the detail callout was created. If the callout object was created on a design layer, select the callout and edit it directly. If the callout object was created within a viewport's annotation space, select the viewport and edit its annotations (see "Editing a Detail Callout in a Viewport" on page 1659).

If **Use Automatic Drawing Coordination** is selected in document preferences, a change to the **Drawing Number** field for the detail viewport automatically changes the field for the viewport's drawing label, and vice versa. Additionally, if **Text Auto-Fill** is selected for the detail callout, a change to the **Drawing Number** field for the viewport automatically changes the field for the detail callout.

There are several ways to change the appearance of a detail callout object.

- Use the Attributes palette to change the crop object's line color, thickness, or type
- Use the Object Info palette to change the object properties (see "Properties of Detail Callouts" on page 1642)
- Use the **Text** menu to change the attributes of the marker text (or apply a text style to it)
- Use the **Selection** tool to adjust the marker position

You can also adjust the shape or location of the callout object, which will change the contents of the detail viewport associated with the callout. If there are multiple instances of the detail callout, all instances are updated.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Use the **Selection** tool to move the entire callout
- Use the **Reshape** tool to reshape the callout object

Editing a Detail Callout in a Viewport

If the callout was created from a viewport, the callout object cannot be edited directly because it is in the viewport's annotation space.

To edit a detail callout object in a viewport:

1. Select the original viewport and then select **Modify > Edit Viewport**. The Edit Viewport dialog box opens.
2. Click **Annotations** and then click **OK** to enter Edit Annotation mode.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a viewport and select **Edit Annotations** from the context menu.

   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Viewport** command becomes available from the **Modify** menu, and the **Exit Viewport Annotation** button is visible in the top right corner of the drawing window.
3. Select the detail callout and edit it as needed. The parameters in the Object Info palette are described in "Properties of Detail Callouts" on page 1642.
4. Click **Exit Viewport Annotation** to exit Edit Annotation mode and return to the sheet layer.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports
Reshaping Objects

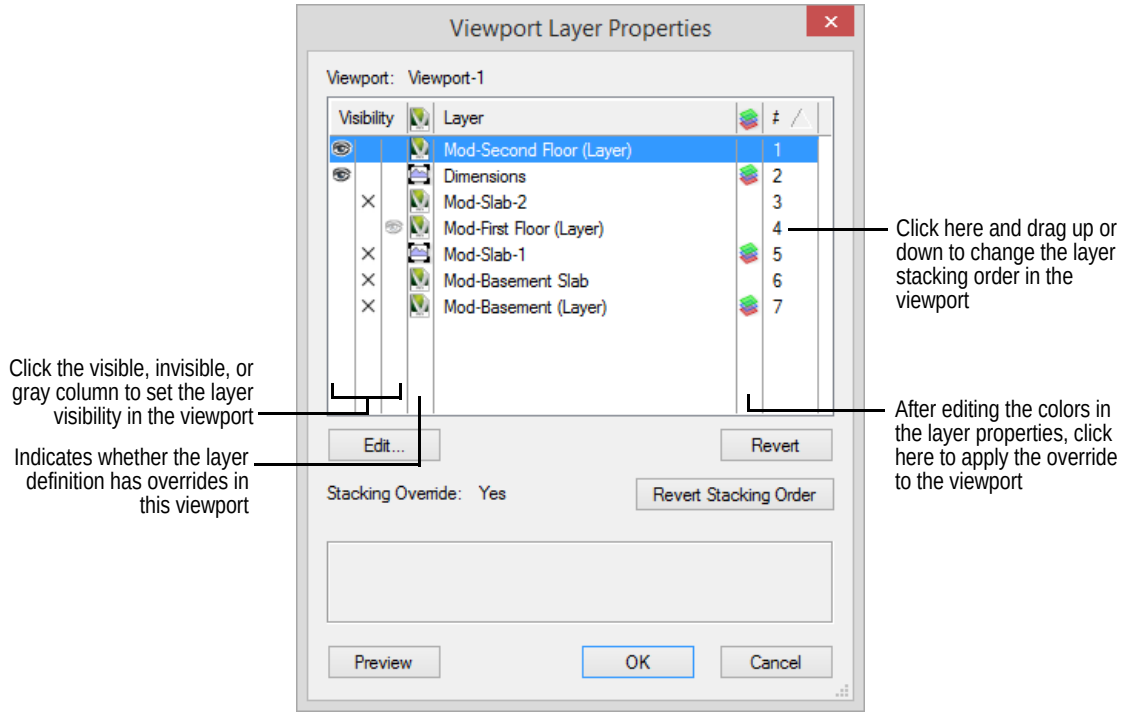
Changing the Layer Properties of Sheet Layer or Design Layer Viewports

The viewport's layer visibility, opacity, stacking order, and colors can be changed from the sheet layer or the design layer (Vectorworks Design Series required). Other viewports, as well as the design layer properties, are not affected. The viewport attributes can be tailored as desired for presentation; several copies of the same viewport can appear completely different.

To change the viewport layer properties:

1. Select the viewport.
2. From the Object Info palette, click **Layers**.

   The Viewport Layer Properties dialog box opens.Change layer visibilities and/or make layer attribute overrides for the selected viewport.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click to show/hide the parameters.

Parameter	Description
Layer list	Lists the viewport layers and their visibility, edited status, layer color use status, and stacking order. Click the triangle in the heading of an active column to toggle between ascending and descending sort order based on that column parameter.
Visibility	Click in a layer visibility column to change the layer visibility for this viewport. • Column 1: Visible (displays objects in this layer) • Column 2: Invisible (hides objects in this layer) • Column 3: Gray (displays objects in this layer as dimmed)
(Source) 	• Column 4 indicates whether the layer definition is the same as the source layer, or the layer definition has overrides in this viewport. When a layer is edited with the **Edit** button, the override icon displays automatically. Click the **Revert** button to revert to the original settings. The layer definition is from the viewport's source layer   Layer overrides exist in this viewport
(Layer Colors) 	Click to apply the viewport layer colors set in the Edit Viewport Design Layers dialog box (click **Edit** to set the colors, as described in the next step), overriding the design layer colors. This setting is independent of the **Use layer colors** document preference.
# (Stacking Order)	Displays the layer stacking order; drag a layer within the # column to change its stacking order. This column displays only when the viewport is in Top/Plan view.
Edit	Opens the Edit Viewport Design Layers dialog box, to override the properties of the selected layer
Revert	Returns the settings in the Edit Viewport Design Layers dialog box to their default values and removes the override icon in the **Source** column

Parameter	Description
Stacking Override	Indicates whether the layer stacking order in the viewport is different from the design layer stacking order. Click **Revert Stacking Order** to return to the original design layer stacking order.
Use embedded design layer viewport settings for (Vectorworks Design Series required)	• **Layer overrides**: If the sheet layer viewport contains a non-referenced design layer viewport for which layer overrides have been set, this option uses the design layer viewport overrides, ignoring any layer overrides that may be set here for the sheet layer viewport. • **Layer visibilities**: If the sheet layer viewport contains a non-referenced design layer viewport for which layer visibilities have been set, this option uses the design layer viewport visibilities, ignoring any layer visibilities that may be set here for the sheet layer viewport. Referenced design layer viewports are not affected by either of these settings.
Preview	Click to preview the layer property settings in the selected viewport

3. To override the layer properties (for viewport display), select one or more viewport layers and click **Edit**.

   Alternatively, double-click on a viewport layer to edit it.

   The Edit Viewport Design Layers dialog box opens.

4. The same parameters apply when you create a design layer (see “Setting Design Layer Properties” on page 165); for viewport layers, only the stacking order, transfer mode or opacity, and colors can be edited. These edits apply to the current viewport only, though they can be transferred to other viewports with the **Eyedropper** tool.

   The viewport layer colors can be controlled separately from the design layer colors, for flexible presentation output. Click **Colors** to override the fill and pen colors for the selected viewport layer. To see the effects of the color override, **Use Layer Colors** must be selected in the Viewport Layer Properties dialog box for the selected viewport. This is similar to the way that **Use Layer Colors** must be selected in document preferences to see the layer color settings for a design layer, as described in “Setting the Design Layer Color” on page 170.

5. Click **OK** to return to the Viewport Layer Properties dialog box.

   Click **Preview** to evaluate the results of the property changes.

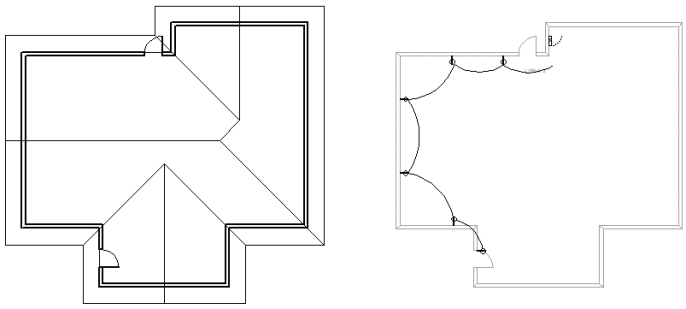
6. Click **OK** to return to the sheet layer or design layer.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Changing the Class Properties of Sheet Layer or Design Layer Viewports
Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports
Reshaping Objects
List Box Functionality

Changing the Class Properties of Sheet Layer or Design Layer Viewports

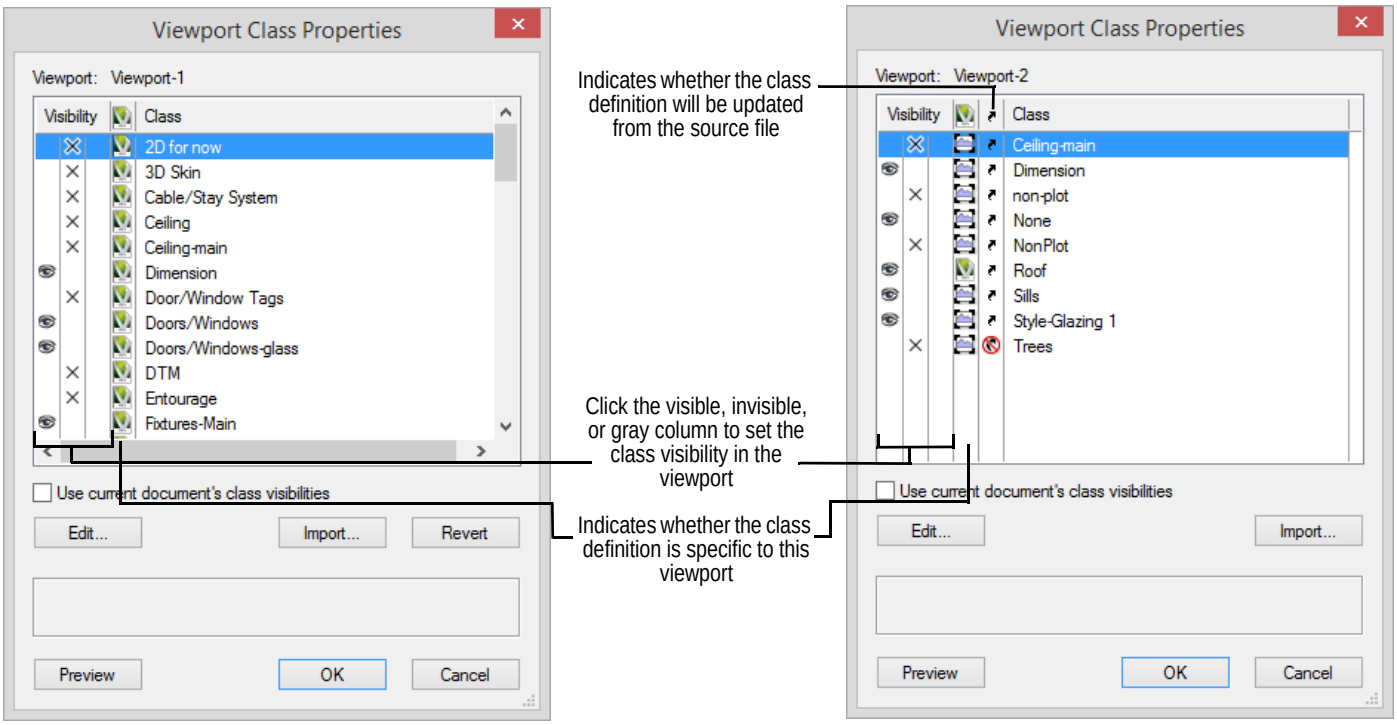
The class visibilities and attributes of a selected viewport can be changed from the sheet layer or the design layer. This does not change the class properties or the class visibility for the original design layers or for other viewports. The viewport attributes can be tailored as desired for presentation; several copies of the same viewport can appear completely different.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



To override viewport class properties:

1. Select the viewport.
2. From the Object Info palette, click **Classes**.

The Viewport Class Properties dialog box opens. The dialog box functionality is slightly different for referenced (Vectorworks Design Series required) and non-referenced (internal) viewports. Change class visibilities and/or make class attribute overrides for the selected viewport.



Non-referenced Viewport

Referenced Viewport

Click to show/hide the parameters.

Parameter	Description
Class list	Lists the viewport classes and their visibility and edited status; click in a class visibility column to change the class visibility for this viewport. Click the triangle in an active column to toggle between ascending and descending sort order based on that column parameter.

Parameter	Description
Visibility	Click in a class visibility column to change the class visibility for this viewport. • Column 1: Visible (displays objects in this class) • Column 2: Invisible (hides objects in this class) • Column 3: Gray (displays objects in this class as dimmed)
(Source) 	Column 4 indicates whether the class definition is the same as the source class, or the class definition has overrides specific to this viewport. When a class is edited with the **Edit** button, the viewport-specific icon displays automatically. Click the **Revert** button to revert to the original settings. The class definition is from the source class   The class definition is specific to this viewport • **Non-referenced viewports:** All classes use the document class definition by default. To change the appearance of a class in this viewport, click **Edit** and change the settings as needed; the viewport-specific icon displays automatically. Click the **Revert** button to revert to the original settings. • **Referenced viewport:** (Vectorworks Design Series required) Because they come from an external file, all classes show as viewport-specific by default. If the same class exists in both the viewport and in the current file, they remain completely separate. By default, the class definition in the viewport will be updated from the source file when the reference is updated.  To change the appearance of a class for this viewport, leave the **Source** set as viewport-specific; then click **Edit** and change the class settings as needed. Also set the class to not update from the source file; otherwise, the edits will be removed the next time the reference is updated.  To use the same class definition both in the current file and in the viewport, click the **Source** icon to toggle the setting. If the viewport class does not exist in the current file, it will be imported. If the class is also set to update from the source file (), an alert message prompts you to verify that you want to reference the class definition. If you answer Yes, the class in the current file and in the viewport will be changed whenever the reference to the viewport source file is updated. If you answer No, the **Source** reverts to the default setting.
(Update)  (Referenced viewports only)	Column 5 indicates whether the class definition will be updated from the source file when the reference is updated. Click the icon to toggle the setting. Update Do not update
Use current document's class visibilities (Vectorworks Design Series required)	When enabled, the viewport uses the current document's class visibilities. The option is automatically enabled when layer links or referenced layers are converted into viewports; it is disabled by default for new design layer viewports.
Edit	Opens the Edit Class(es) dialog box, to make overrides to the selected class that only apply to the current viewport (see "Setting Class Properties" on page 179); drawing objects' class setting should be "By class" to take advantage of class overrides

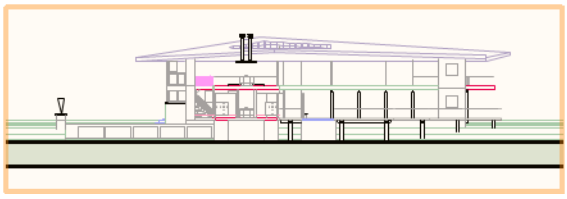
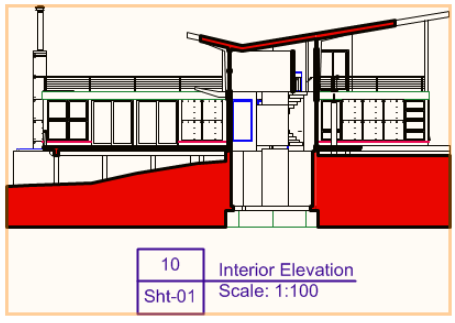
Parameter	Description
Import	Opens the Import Attributes dialog box, to import the class attribute settings from the file. The attributes can be imported for the classes selected in the Viewport Class Properties dialog box, from corresponding classes, or from a specified class in the file or a specified viewport. Click **OK** to import the class attributes into the selected viewport. (The **Eyedropper** tool can also transfer class override attributes between viewports.)
Revert	Sets the selected class back to its original document attributes, undoing any class overrides; also removes the override icon in the **Source** column
Use embedded design layer viewport settings for (Vectorworks Design Series required)	• **Class overrides**: If the sheet layer viewport contains a design layer viewport for which class overrides have been set, this option uses the design layer viewport overrides, ignoring any class overrides that may be set here for the sheet layer viewport. • **Class visibilities**: If the sheet layer viewport contains a design layer viewport for which class visibilities have been set, this option uses the design layer viewport visibilities, ignoring any class visibilities that may be set here for the sheet layer viewport
Preview	Click to preview the class visibility and attribute settings in the selected viewport

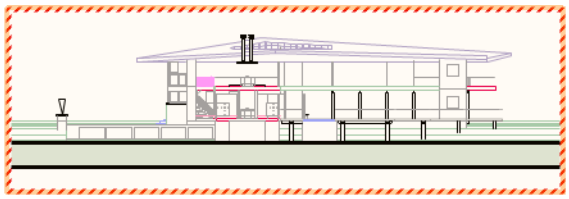
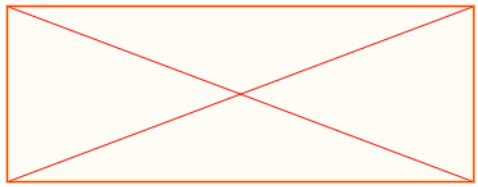
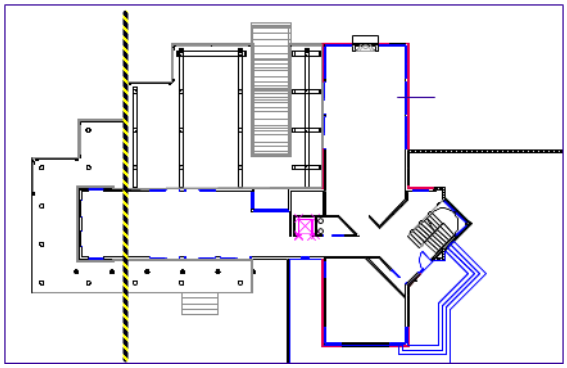
3. Click **OK** to apply the class visibility and attribute changes to the selected viewport.

Changing the Layer Properties of Sheet Layer or Design Layer Viewports
Modifying Viewports
Viewport Properties
Viewport Status
Updating Viewports
Reshaping Objects
List Box Functionality

Viewport Status

The status of a viewport or section line is indicated visually.

Viewport Status	Description
Normal	A normal, up-to-date viewport displays with orange highlighting when selected  

Viewport Status	Description
Out of date	When the objects in a viewport have changed since the viewport was created or last updated, the viewport becomes out of date. An out-of-date viewport is indicated by red text on the **Update** button on the viewport's Object Info palette. Optionally, an out-of-date viewport also displays with a red and white striped border around the viewport (see "Document Display Preferences" on page 60).  
Empty	A viewport displays as a red "X" when the associated design layer contains no objects or the objects are hidden, or when the associated design layer is set to "invisible" 
Unlinked (section line) (Vectorworks Design Series required)	An unlinked section line (disconnected from its associated section viewport, possibly because the section line was pasted from a copy, duplicated, or mirrored) displays as a black and yellow line, and "Not Linked" is displayed in the Object Info palette 

Updating Viewports
Modifying Viewports
Viewport Properties

Updating Viewports

Changes that affect the appearance of a viewport are automatically updated for a wireframe viewport. However, if changes require the viewport to be rendered again, the viewport will be displayed as an out-of-date viewport.

> If a sheet layer with an out-of-date viewport is printed, a message prompts you to either print the viewport as an out-of-date viewport or update the viewport(s) on the sheet layer before printing.

Rendered viewport updates occur in the background for Renderworks render modes; you can continue working in the file while the viewports are updating. See "Background Rendering" on page 1594.

Updating Selected Viewports

To update selected viewports:

1. Select the viewport(s).
2. On the Object Info palette, click **Update**. Alternatively, select **View > Update Selected Viewports**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on the viewport and select **Update** from the context menu.
3. The selected viewports are updated.

Updating All Viewports

To update all the viewports in the file:

1. Select **View > Update All Viewports**.
2. All viewports on all sheet layers are updated.

Canceling Viewport Updates

To cancel viewport updates:

To cancel the updates of all viewports, including those queued to update for background rendering:

1. Select **View > Cancel All Viewport Updates**.
2. The updates are canceled.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Viewport Status
Modifying Viewports
Viewport Properties
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing and Exporting Files

The Vectorworks program has the ability to import and export to a variety of file formats. Your work may require you to import drawings from colleagues that use previous versions of Vectorworks or other programs. You may also have to export your drawings for those colleagues. The import and export functionality provides the means to incorporate Vectorworks files with other program's files, including other CAD programs, worksheet, and word processing programs.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Drag and Drop Importing

In addition to the import commands on the **File** menu, a drag and drop import shortcut is available for most file formats. Click the file to import and drag it into a window where a Vectorworks document is open.

Drag and drop importing is supported for the following file types:

- 3DS (.3ds)
- DXF/DWG (.dxf, .dwg)
- EPSF (.eps)
- IFC (.ifc, .ifcxml, .ifczip)
- IGES (.igs)
- Image files (various file extensions)
- Metafile (.emf, Windows only)
- Parasolid (.x_t)
- PDF (.pdf)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- PICT (.pct, .pict)
- SAT (.sat)
- Shapefile (.shp)
- SketchUp (.skp)

The imported objects will be centered on the drop point, except for the following file types: DXF/DWG, IFC, Shapefile, and SketchUp.

D Importing in PartSpec Format

The **Import PartSpec** command imports Thomas Register PartSpec files saved in either DWG/DXF (2D objects) or IGES or ACIS (3D objects) format. It places all parts in a single class on the active design layer and deletes any extraneous layers. The **Import PartSpec** command does not differentiate between a file created in the Thomas Register and other programs.

To import PartSpec files:

1. Select the **Import PartSpec** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Import PartSpec**
   - Landmark workspace: **Landmark > Machine Design > Import PartSpec**
   - Spotlight workspace: **Spotlight > Machine Design > Import PartSpec**
2. A message indicates that the undo cache will be cleared, and that this action cannot be undone; click **OK** to proceed.

   The Import Thomas Register PartSpec dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Import 2D Part (DWG/DXF)	Select to import a 2D part from a DWG or DXF file
Import as symbol named	Select to import the part as a symbol; enter the name of the symbol in the symbol name field
Import as group	Select to import the part as a group
Attach part information to the group or symbol	Select to attach the manufacturer's part information to the symbol or group. The information is attached through the Part Information record format which is created if not already existing in the file.
Put objects into class	Select the class to apply to all objects being imported
Import 3D Part	Select to import a 3D part
ACIS	Select to import the part from an ACIS file
IGES	Select to import the part from an IGES file

3. Select the desired import options and click **OK**.

Import	Description
2D part from either a DWG or DXF file	In the Import DXF/DWG Files dialog box, select the file to import, and then click **Open**. The DXF DWG Import Options dialog box opens. (See "DXF/DWG and DWF File Import" on page 1716). The only setting that may need to be edited in the DXF DWG Import Options dialog is **Units Setting In File** under the Primary Settings tab. This setting refers to the units in the PartSpec DXF or DWG file, not the units of the current Vectorworks file. Do not edit the **Import DXF Layers As Classes** field under the Graphics Attributes tab or the **Group Record Fields** under the Objects tab. Changing these options will prevent objects from importing correctly. Depending on the imported part, the Map Colors to Line Weights dialog box and/or the Font Mapping dialog box may open. Make any necessary changes and then click **OK**. The part is imported into the file based on the specified settings.
3D part from an ACIS or IGES file	If this is an IGES file, the Import IGES File dialog box opens. Select the file to import, and then click **Open**. If this is an ACIS file, the Import SAT File dialog box opens. Select the file to import, and then click **Open**. The part is imported into the current class and design layer, and is available for editing and/or converting into a symbol.

Importing in SketchUp Format

The **Import SketchUp** command allows architectural drawings created in SketchUp® (versions 4 and above) to be imported into a Vectorworks file. SketchUp component instances are imported as 3D symbols, and geometry can be designated as walls, roof faces, and floors. After importing, refine the design using Vectorworks editing tools, or replace imported 3D symbols with Vectorworks 3D symbols.

Flexible import options either allow automatic conversion of architectural elements based on their orientation, or allow geometry conversion to be mapped to specific SketchUp materials or layers.

Consider the following points when importing a SketchUp file into a Vectorworks file.

- SketchUp layers are imported as classes in Vectorworks. Geometry from invisible layers in SketchUp have their class attributes set to invisible in Vectorworks.
- SketchUp groups are imported as Vectorworks group objects. If the SketchUp group has a scale assigned, the imported objects inside the group will be transformed with that scale.
- SketchUp components are imported as Vectorworks 3D symbols except for the rare cases when they have non-linear scale, in which case they are imported as groups of transformed geometry.
- A component which creates an opening in a vertical face is imported as a symbol in wall, if attached to an object imported as a Vectorworks wall.
- When selected, architectural objects in Vectorworks are created from SketchUp faces that are not contained in SketchUp groups of components.
- The SketchUp format supports materials on the front and back of each face of an object. When importing as 3D polygons, only the front material is assigned to the imported objects, as Vectorworks only supports one material for 3D polygons.
  - When importing as architectural objects, both materials are imported and assigned to the imported objects, as Vectorworks supports several materials in this case.
- When importing as architectural objects, faces which are not part of a group or a component instance in SketchUp will import as roofs, slabs, or floors.

- Walls are unjoined upon import, and may need to be rejoined.

To import a SketchUp file:

1. Select **File > Import > Import SketchUp**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.
2. The Import SketchUp Model dialog box opens. Select the file to import, and click **Open**.
3. The SketchUp Import Settings dialog box opens.
4. Select the method for importing geometry through the options on the Import Method tab.

Click to show/hide the parameters.

Parameter	Description
Import Method	
Simple	Select to automatically import all objects in the input file. • Meshes/3D Polygons will import all objects as meshes or 3D polygons. • Architectural Objects will import objects as Vectorworks architectural objects. Vertical faces will become walls, horizontal faces will become slabs, and oblique faces will become roof faces. Vectorworks recognizes faces in the SketchUp file and determines if they should be converted to building elements by analyzing the angle of the face relative to the horizontal plane.
Custom	Select to specify which materials/layers to import. All materials/layers are selected for import by default.
Import By	Select Layer to display the list of layers in the SketchUp file or Materials to display the list of materials. A check mark to the left of each layer/material indicates that it is to be imported. To omit one or more layers/materials, select the item(s) to be omitted and click to remove the check mark.
Layer/Material Import Method	Select the material/layer from the list and specify whether it should be imported as either a mesh/3D polygon or as an architectural object.

5. Click the Options tab to specify which material properties to import, whether to import geometry as mesh objects or 3D polygons, and set the parameters for architectural objects.

Click to show/hide the parameters.

Parameter	Description
Options	
Import Materials	Select whether to assign fill colors from the imported objects according to the colors of the materials of objects when selected. If **Layer/Material Import Method** is selected, any imported architectural objects will automatically be assigned the correct properties as defined in the SketchUp file.
Import Transparency and Image Data as Renderworks Textures	Select to import texture and transparency materials as Renderworks texture resources

Parameter	Description
Create Renderworks Textures for All Materials	Select to create Renderworks textures for all imported materials from the input file. Solid colors will also be imported as Renderworks textures.
Import the Geometry from the File as	Select whether to import geometry as a group of mesh objects or 3D polygons
Vectorworks Mesh Object	Select to import all geometry as a group of mesh objects. When importing as mesh objects with materials, the SketchUp faces will be triangulated, this affects the appearance of the imported file if pen color is used to display the face edges. Vectorworks maintains the original mesh smoothing normal and texture mapping data from SketchUp. This information can be edited on the Render tab of the Object Info palette, though some of the imported rendering information may be lost during editing.
Groups of 3D Polygons	Select to represent all imported geometry as a group of 3D polygons. When importing as 3D polygons, the SketchUp faces will not be triangulated except when importing complex faces with textures. SketchUp defines a complex face as a face containing either inner loops or openings cut by attached component instances or attached groups.
Preserve Texture Assignment and Mapping	If **Import Materials** is selected, and you are importing geometry as polygons, specify whether textures and texture mappings are applied to imported 3D polygons to preserve their overall appearance (Renderworks required).
Architectural Objects parameters when created from automatic or layer/material assigned import method	Specify the thickness at which to create various building elements: • **Wall Thickness**: Set the thickness to assign to wall geometry during import; this value must be greater than zero • **Floor Thickness**: Set the thickness to assign to floor geometry during import • **Roof Face Thickness**: Set the thickness to assign to roof face geometry during import

6. Click **OK**.
7. SketchUp geometry and components are imported as specified, and are automatically selected. The Resource Browser is populated with the converted 3D symbols.

   Items are imported at a 1:1 scale. It may be helpful to click **Fit to Objects** on the View bar to zoom to fit the objects in the drawing. It may also be helpful to adjust the layer scale at this time.

Importing in PICT Format

This section only applies to Mac users.

PICT files were first produced by MacDraw and other object-oriented drawing programs. When you import a PICT file with the **Import PICT** command, multiple bitmap objects may be created, which can be edited individually later on. When you import a PICT file with the **Import PICT as Bitmap** command, the image is imported as a single bitmap object. The **Trim**, **Clip**, and **Shear** tools cannot be used on the imported bitmap. Bitmap objects imported with these commands automatically have PNG compression applied.

To import a PICT file:

1. Select **File > Import > Import PICT** or **File > Import > Import PICT as Bitmap**.

The Import PICT File dialog box opens.

2. Select the file to import, and click **Open**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

Exporting as a Database

The **Export Database** command exports all records of a particular format as a file that can be used in a database program, such as FileMaker Pro and Microsoft Access. The Vectorworks program provides a variety of formats to select from when exporting records as a database, including comma-delimited, tab-delimited, merge, DIF, and SYLK.

To export a file from a database:

1. Select **File > Export > Export Database**.

   The Export Database dialog box opens.

2. Select the database and the file format to export, and click **OK**.
3. Specify the export file name and destination, and click **Save**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Database Connectivity

R Exporting a Rendered Drawing

All files rendered with the Renderworks product can be exported. It is possible to export all or part of a drawing depending on the selected export format.

The image is re-rendered automatically before export.

The **Export Image File** command exports a rendered drawing to a variety of image file formats. This command is described in detail in “Exporting an Image File” on page 1683.

The following additional export options are available only as part of the Renderworks product.

- “High Dynamic Range Image (HDRI) Export” on page 1672
- “QuickTime VR Object Export” on page 1674
- “QuickTime VR Panorama Export” on page 1675

R High Dynamic Range Image (HDRI) Export

When the Renderworks product is installed, the HDRI option is added to the list of export file choices. This command creates an HDR file. This is a good choice for export when touching-up the image in an imaging program; the exported image is “lossless” and will not have quantization artifacts when color or exposure is further adjusted after export.

To export a file in HDR format:

1. Select the Renderworks rendering mode to use for the export, and render the image.
2. Select **File > Export > Export High Dynamic Range Image (HDRI)**.

   The Export Image File dialog box opens. Specify the dimensions and resolution for the exported image.

Click to show/hide the parameters.

Parameter	Description
Export Area	
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
All Visible Objects	Exports an image that includes all visible objects (objects do not have to be currently on screen to be considered visible)
Current View	Exports an image that is exactly as it displays on the current screen
All Pages as Single Image	Not applicable for HDRI export
Each Page as Separate Image	Not applicable for HDRI export
Marquee	Exports the portion of the image specified with a marquee box. Select this option and then click **Draw Marquee** to temporarily close the dialog box. Click, and then drag to specify the area for export; the marquee dimensions are displayed on the Data bar. Click to set the export area and return to the Export Image File dialog box. The **Pixel Dimensions** of the image are automatically set to the marquee dimensions.
Dimensions	
Lock Aspect Ratio	Select to maintain the image aspect ratio when specifying dimensions
Resolution	Specifies the printed image resolution in pixels per inch
Pixel Dimensions	
Width/Height	Specifies the exported image dimensions; if **Lock Aspect Ratio** is selected, changes to one dimension will update the other to maintain the aspect ratio
Print Size	
Width/Height	Specifies the printed image dimensions in the selected **Unit**; if **Lock Aspect Ratio** is selected, changes to one dimension will update the other to maintain the aspect ratio
Units	Select a unit to apply to the **Print Size** parameters
Preview	Displays a rendered or wireframe preview according to the current settings
Render	Updates the preview with a rendered view using the currently set rendering option
Wireframe	Updates the preview with a wireframe view
Memory Required/Estimated File Size	Not applicable for HDRI export
Update	Not applicable for HDRI export
Format	
File Type	Not applicable for HDRI export (always HDR format)
Compression	Not applicable for HDRI export
Update visible out of date viewports prior to exporting	Automatically updates any visible, out-of-date viewports before exporting
Reset all plug-in objects that require a reset prior to exporting	Automatically resets plug-in objects that require an update (such as data stamps) before exporting

Parameter	Description
Export Georeferencing File (Vectorworks Architect or Landmark required)	Available only if the current design layer is georeferenced. Along with the image file, exports a worldfile that describes the exported image's location, scale, and rotation in the geographic coordinate system. If multiple georeferenced layers are visible, the worldfile will be written according to the topmost layer. See "GIS and Georeferencing" on page 775 for more information.

3. Click **Save**.

   The Export OpenHDR File dialog box opens.

4. Specify the export file name and destination, and then click **Save**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Exporting a Rendered Drawing

QuickTime VR Object Export

When the Renderworks product is installed, the QuickTime® Virtual Reality object option is added to the list of export file choices. This command creates a QuickTime VR object file of the selected object, which can then be opened in QuickTime. Movie resolution is fixed and does not depend on the size of the window in the Vectorworks file; the current selection determines the zoom and center for the movie, set so that the selected object is entirely visible and centered.

To obtain more control over the zoom and view center, create an "invisible" 3D object with no fill or pen style. Select the object before exporting the movie.

QuickTime must be installed to view or create QuickTime movies; it is a separate program. QuickTime includes a viewer for opening several different file types.

To export an object as a QuickTime VR object:

1. Render the drawing with a Renderworks rendering mode, which is also used for the movie.

   Due to a limitation, Renderworks backgrounds cannot be exported.

2. Select the object to export.
3. Select **File > Export > Export QuickTime VR Object**.

   The QTVR Object Options dialog box opens.

4. Select the number of frames and spin options for creating the VR object file, and then click **OK**.

Click to show/hide the parameters.

Parameter	Description
Frames	Specify how many frames to create in the file; more frames take longer, but increase the quality of the exported file. **Total Frames** displays how many frames will be created based on the horizontal and vertical frames specified.
Horizontal	Specifies the number of horizontal frames to create
Vertical	Specifies the number of vertical frames to create
Spin	Specify the angular sweep desired, relative to the front view of the object
Left	Indicates the left pan angle (0 to 180 degrees)
Right	Indicates the right pan angle (0 to 180 degrees)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Above	Indicates the angle above the horizon (0 to 90 degrees)
Below	Indicates the angle below the horizon (0 to 90 degrees)

5. Specify the .mov file name and location, and click **Save** to generate the QuickTime movie.

Exporting a Rendered Drawing

QuickTime VR Panorama Export

When the Renderworks product is installed, the QuickTime Virtual Reality Panorama option is added to the list of export file choices. This command creates a QuickTime VR panorama file of the drawing, which can then be opened in QuickTime. Movie resolution is fixed and does not depend on the size of the window in the Vectorworks file; the current Vectorworks view (XYZ coordinates of the view origin or, in other words, the center of projection) becomes the pivot point for the QTVR panorama view, as if the view was rotated to be flat, and then oriented towards the positive Y axis.

QuickTime must be installed to view or create QuickTime movies; it is a separate program. QuickTime includes a viewer for opening several different file types.

To export a drawing as a QuickTime panorama:

1. Render the drawing in **Perspective** projection with a Renderworks render mode, which is also used for the movie.

   Due to a limitation, Renderworks backgrounds cannot be exported.
2. Select **File > Export > Export QuickTime VR Panorama**.
3. Specify the .mov file name and location, and click **Save** to generate the QuickTime movie.

Exporting a Rendered Drawing

Exporting in CINEMA 4D Format (3D only)

Exports 3D objects in the unified view or active layer to the .c4d format, for use in CINEMA 4D. A “Vectorworks Scene” is created within the CINEMA 4D project. Vectorworks layers are exported as container objects in the scene hierarchy. Vectorworks classes become CINEMA 4D layers, allowing visibility to be controlled across the scene. Layer names, object names, and object types are preserved.

Exporting a File to CINEMA 4D

To export a CINEMA 4D file:

1. Select **File > Export > Export CINEMA 4D (3D only)**.
2. Specify the export file name and destination, and then click **Save**.

Sending a File Directly to CINEMA 4D

An additional command allows you to send a file to CINEMA 4D directly, when CINEMA 4D is installed.

To send a file to CINEMA 4D:

1. Select **File > Send to CINEMA 4D (3D only)**.

2. For CINEMA 4D version 12 or later, this exports the file to .c4d format (3D objects only), launches the CINEMA 4D application, and opens the file. During export, the Update Merge Options dialog box opens. Specify whether to create a new project or merge into the current one. By default, the options are set so that the geometry from the Vectorworks program is merged, but material and sky objects in CINEMA 4D are unaffected.

   Once the file has been sent to CINEMA 4D, select the **Save Project with Assets** command in CINEMA 4D. This keeps the texture assets with the .c4d file, and prevents them from becoming disconnected in CINEMA 4D.

   For CINEMA 4D versions prior to R13.053, the Send to CINEMA 4D dialog box opens. Select whether to create a new scene or merge into the current scene.

R Exporting in COLLADA Format (3D only)

This command exports 3D objects in the unified view or active layer to .dae format. When saved, textures are placed in a "tex" folder in the same location as the .dae file. This format may provide better results than the older 3DS file format.

To export a COLLADA file:

1. Select **File > Export > Export COLLADA (3D only)**.
2. Specify the export file name and destination, and then click **Save**.

R Exporting in FBX Format (3D only)

This commend exports 3D objects in the unified view or active layer to .fbx format. This format may provide better results than the older 3DS file format.

To export a FBX file:

1. Select **File > Export > Export FBX (3D only)**.
2. Specify the export file name and destination, and then click **Save**.

D Exporting in KML Format

The **Export KML** command exports 3D drawing components to KML, the markup language used by the Google Earth application. The resulting file can then be opened in Google Earth, in order to provide a visualization of how a building fits into its intended environment, for example. KML models can also be uploaded and shared through the Trimble ® 3D Warehouse.

To export a KML file:

1. Select **File > Export > Export KML (3D only)**.
2. The KML Export Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Mapping	A 2D reference point on the object is mapped to specific Google Earth coordinates during export. If the file is georeferenced, the following mapping fields are filled in automatically and disabled.

Parameter	Description
Set Reference Point to Next Mouse Click/Set Reference Point to	To specify the reference point manually, select **Set Reference Point to Next Mouse Click**. Alternatively, to specify X and Y coordinates for the reference point, select **Set Reference Point to** and enter the appropriate values.
Latitude	Enter the latitude where the model will be positioned in Google Earth, using DMS notation (for example, 39d 12m 34s N)
Longitude	Enter the longitude where the model will be positioned in Google Earth, using DMS notation (for example, 76d 51m 44s W)
Page North	Enter the angle between the drawing's Y axis and true north in DMS notation; positive angles proceed clockwise
Z Height Offset	Enter the difference between the Vectorworks active layer plane and the Google Earth ground plane
Objects To Export	Select the specific object types to be exported; leaving out unnecessary objects can reduce the KML file's size
Layer(s) to export from	Select the design layers to export 3D objects from—either the active layer only, or all visible layers. To export all design layers, first set the layer options to show all layers. If multiple layers are exported, in Google Earth, the Vectorworks layers are listed in the Places panel on the left side of the application window, and their visibilities can be controlled individually. To export some (but not all) layers, create a new design layer from which to export. On the new layer, create a design layer viewport (or create layer links) that shows only the layers to export. Then export that layer only. With this method, only one layer displays in Google Earth.

3. Select the export parameters, and then click **OK**.
4. Specify the export file name and destination, and then click **Save**.

Exporting in STL Format

The **Export STL (3D Only)** command exports 3D surfaces and solids to an STL (Stereo Lithography) file. Once exported, this file can be used for rapid prototyping purposes or imported into other CAD packages.

To export an STL file:

1. If 3D objects from only one layer should be exported, make that layer active. If only selected objects should be exported, select the objects.
2. Select **File > Export > Export STL (3D only)**.
3. The Export STL Options dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Export	
Selected Objects on Active Layer	Exports only selected 3D objects from the active layer

Parameter	Description
All Visible Objects on Active Layer	Exports all visible 3D objects from the active layer (including objects in design layer viewports)
All Visible Objects on All Layers	Exports all visible 3D objects from all layers in the Vectorworks file (including objects in design layer viewports)
File Type	Select **ASCII** or **Binary** for the file type
Export Quality (Number of Polygons)	Controls the conversion quality of 3D objects that have rounded sides. Move the slider bar left to convert rounded objects using fewer large polygons (thus lower quality). Move the bar right to create rounded objects of higher quality. Exporting more polygons slows down the export and increases the size of the output file.

4. Select the export options, and then click **OK**.
5. Specify the export file name and destination, and then click **Save**.

Click here for a video tip about this topic (Internet access required).

Exporting to an Earlier Version of the Vectorworks Program

The Vectorworks software can save a copy of the file in a format that can be opened and manipulated in an older version of the Vectorworks program. Features and capabilities of the current version may not be available in previous versions of the program.

1. Select **File > Export > Export as Vectorworks <<version>> File**.
2. Specify the export file name and destination, and then click **Save**.

Exporting in gbXML Format

The gbXML file format transmits building information for energy analysis.

Valid Vectorworks Architect objects that are included in gbXML export are spaces, walls, slabs, windows, doors, columns, and roofs. Geo-referenced site location, if any, is included. The model must have a structure with stories, even if the model is very simple with only one design layer per story. Columns to be included in the export must be along the periphery of a space; columns totally within or outside of a space are not considered. All space objects must be fully enclosed by walls, slabs, and/or roof elements, and there should be a full-floor boundary for each story in the building. The **Place Spaces** command makes this simple; it places the required spaces automatically and ensures that they properly touch their bounding objects.

The model can include shading surfaces (overhangs, awnings, or other surfaces, including walls roofs, and slabs, that do not bound a space and lie outside the bounds of the model).

For best results, before exporting, simplify the model so that only objects that affect energy consumption are visible. Eliminate, or make invisible, items that do not contribute to the energy profile of the building, such as unnecessary geometric detail, furniture, and fixtures. (Note that spaces in invisible classes are ignored.)

Automatically Creating Spaces

For export, the model must contain space objects, which represent the essential organizational elements for gbXML exchange. Spaces must be contained by walls, slabs, and roofs. If the model does not contain space objects for each enclosed area and the full floor of each story, the **Place Spaces** command creates the necessary spaces for export, with full boundaries on all sides so that the exporter can properly determine the space volume.

To create spaces optimized for gbXML export:

1. Select **AEC > gbXML > Place Spaces**.

   The Place Spaces dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Place Room Spaces in Class	Creates spaces automatically for the enclosed room areas of the model; select a class for the spaces, or select New to create a new class
Place Full-floor Spaces in Class	Creates a full-floor space automatically within the boundary of the exterior walls on each story of the model; select a class for the spaces, or select New to create a new class
Place Spaces for Current Story Only	Places spaces only for the story of the current layer; only this story will be considered for gbXML export
Ignore Small Enclosed Areas	When selected, spaces are not created for small enclosed areas, such as small gaps between walls. This results in more exact export results, but the additional analysis can take some time.

2. Click **OK** to place the spaces automatically.

Exporting to gbXML Format

To export a gbXML-compliant model:

1. Select **AEC > gbXML >Export gbXML**.

   The Export gbXML dialog box opens.

2. Enter the average ground level for the project, and then click **OK**.
3. The gbXML Save File dialog box opens. Type a file name, select a location to save the file, and then click **Save**.

   If the file did not meet the requirements for successful gbXML export, you are notified by alert dialog boxes.

Importing and Exporting in EPSF Format

Importing an EPSF File

EPSF (encapsulated PostScript format) files are produced by many graphics and desktop-publishing programs. EPSFs are typically high-resolution files. On Windows, EPSF files will only display a preview image if a TIFF preview image was selected when the file was created.

To import an EPSF file:

1. Select **File > Import > Import EPSF**.

   The Import EPSF File dialog box opens.

2. Select the file to import, and click **Open**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

Exporting an EPSF File

The **Export EPSF** command exports a drawing to an .EPS file format. EPSF (encapsulated PostScript format) files can be read by many graphics and desktop-publishing programs. The Vectorworks translator exports EPSF files in

Illustrator 88 format. EPSFs contain all drawing elements, and a TIFF preview using standard byte ordering. The Vectorworks program exports these files with high resolution and full accuracy.

To export an EPSF file:

1. Select **File > Export > Export EPSF**.
2. The EPS Preview Options dialog box opens. Specify the type of TIFF preview to export along with the PostScript data, and then click **OK**.
3. The Export as EPSF File dialog box opens. Specify the export file name and destination, and then click **Save**.
4. If any of the objects in the drawing are outside of the print area, a dialog box opens to ask if the print area should be used as the bounds of the EPSF export. If the print area is used as the bounds of the EPSF export, any objects outside of the print area are not included in the export.

Importing and Exporting Image Files

Importing an Image File

Image files, including GIF, JPG, TIF, BMP, and others, can be imported into a drawing directly; for example, you might import a sketch or a logo graphic into a drawing. The Vectorworks Design Series products have an option to create a reference to the original image file, if the original image may change, and you want to keep the imported image up to date.

The following table lists the standard image import formats that are supported.

Export Formats	Mac	Windows
BMP	X	X
GIF	X	X
JPEG	X	X
JPEG2000	X	
PNG	X	X
Photoshop (.psd)	X	
SGI	X	
TGA	X	
TIFF	X	X
ICNS	X	
ICO	X	X
MacPaint	X	
XBitmap	X	
WMPhoto		X

Images are compressed when they are imported into a Vectorworks file, to reduce the file size.

You can also import an image as a resource, to be used as a fill or image-based shader, for example. See “Creating Image Resources” on page 1120 for details.

To import an image file:

1. Select **File > Import > Import Image File**, and then select the image file to import.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

2. The Image Import Options dialog box opens. Specify the options for the imported image.

   Information about the image file is displayed at the top. The compression method that produces the smallest file size is selected by default. Select the method that will give the best balance between compression and detail display. The file size for each type of compression is displayed to help with the selection.

   If the selected option results in a file size larger than the uncompressed size, the image is imported uncompressed.

Compression Method	Description
JPEG	Provides a high amount of compression, resulting in the smallest Vectorworks file size. However, fine detail may be obscured. JPEG compression is most suitable for photographic images.
PNG	Provides a moderate amount of compression, while preserving image details; select **Import as Black and White** to import as a 1-bit monochrome image in PNG format

3. If Vectorworks Design Series products are installed, referencing options display. See “Workgroups and Referencing” on page 207 for more information. Specify the reference options for the imported image as described in the following table.

Click to show/hide the parameters.

Parameter	Description
Reference	References the original image file (master file), so that the imported image can be updated when the master file changes.
Absolute path	Stores the absolute file path of the master file. Select this option when the location of the master file will not change in the future, or if the master file is on another volume.
Path relative to current document	Stores the file path of the master file relative to the target Vectorworks file; this option is available only if the master file is on the same volume as the target file. Select this option if the target file and the master file may be moved to another volume in the future.
Save referenced cache to disk	Saves a copy of the referenced image in the target Vectorworks file; if deselected, the target file is smaller, and the image is automatically updated when the target file is opened
Automatically update reference during file open	Updates the image from the master file each time this file is opened, if the image is out of date

4. Click **OK** to import the image. The Object Info palette displays the image information.

   If Vectorworks Landmark is installed, and the image was georeferenced, a message displays that the image has been translated according to the associated world file. See “Importing and Exporting Georeferenced Raster Images” on page 1702 for more information.

5. Imported images are planar objects and may be assigned to 3D planes. They can be scaled, rotated, and cropped. See “Cropping Imported Images” on page 1682 for more information about cropping.

   The **Trim**, **Clip**, and **Shear** tools cannot be used on an imported image.

   Images imported into the file that are not already in JPEG format can be compressed by JPEG with the **Compress Images** command. See “Compressing Images” on page 1822.

If you use the **Delete Outside Crop** button on the Object Info palette to modify the imported image, the reference will be broken.

A referenced image file can be opened with the default application associated with this file type. To open a referenced image file, use one of the following methods:

- Ctrl-click (Mac) or right-click (Windows) the referenced image in the drawing area, and then select **Open**
- Ctrl-click (Mac) or right-click (Windows) the referenced image file in the Navigation palette, and then select **Open**

Select the referenced image file in the Navigation palette, and then select **Open** from the Navigation menu

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cropping Imported Images
Importing Manual Sketches

Cropping Imported Images

1. Select an imported image, and select **Modify > Edit Crop**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on an image and select **Edit Crop** from the context menu.

   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Bitmap Crop** command becomes available from the **Modify** menu, and the **Exit Bitmap Crop** button is visible in the top right corner of the drawing window.

2. Create a 2D object such as a rectangle, circle, or polyline. The 2D object must define an area; for example, a 2D line cannot be used. Position the 2D object to delimit the new image display area. The pen style can be set from the Attributes palette while in Edit Crop mode. Move and resize the 2D object as needed.
3. Click **Exit Bitmap Crop** to return to the design layer.
4. The cropped image displays; in the Object Info palette, the **Crop** status changes to Yes.

   To change the visibility of the crop object, change the **Crop Visible** setting in the Object Info palette. The crop object can be a visible frame for the bitmap in the drawing, or it can be invisible.

   To permanently delete the part of the image that lies outside the crop object, click **Delete Outside Crop** on the Object Info palette. When prompted, click **Yes** to confirm the deletion. The Object Info palette displays the new image file size.

   If the image file is referenced, another alert message displays that the action will break the reference; click **OK** to confirm the deletion.

   To change, replace, or delete the crop object, select the image and then select **Modify > Edit Crop** to re-enter Edit Crop mode. Alternatively, right-click (Windows) or Ctrl-click (Mac) and select **Edit Crop** from the context menu.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing and Exporting Image Files

Importing Manual Sketches

Scanning and importing sketches, and then tracing them, is a useful technique that can serve as a bridge between manual drawing and CAD. To be useful, sketches for tracing should be drawn to scale and scanned at a relatively high resolution (300 dpi or above).

To import sketches to be traced:

1. After scanning the sketch, save the file in one of the image import formats supported in the Vectorworks program.
2. Determine the scale of the original sketch.
3. In the Vectorworks program, open the file into which the sketch will be imported.

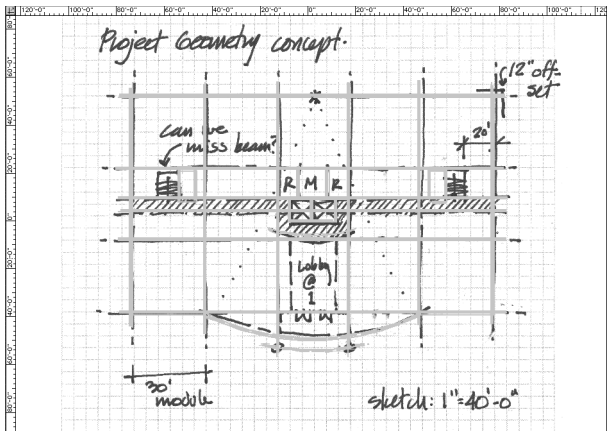
4. Select or create a design layer for importing the sketch. Set the layer to the scale of the sketch. Make this layer the active layer.

   See “Creating Layers” on page 162 and “Setting Design Layer Properties” on page 165.

5. Select **File > Import > Import Image File** to import the sketch.
6. Set the layer back to the desired drawing scale.
7. Move the image to locate it properly. The image can be rotated; however, the best results are obtained when the image is scanned as straight as possible. Consider locking the image once it is in the desired orientation.

   Any manual reshaping or scaling of the image distorts its scale.

The following image shows a scanned sketch used as the basis for a guidelines layout. Draw guidelines in the Guides class. Prior to printing, turn this class off with the **Modify > Guides > Hide Guides** command. See “Guides” on page 71.



Cropping Imported Images

Exporting an Image File

The **Export Image File** command exports a rendered drawing to a variety of file formats, including GIF, JPG, TIF, BMP, WMPhoto, and others. Different portions of the image can be selected for export.

The list of image formats is populated depending upon the operating system and installed drivers.

The following table lists the standard image export formats that are supported.

Export Formats	Mac	Windows
BMP	X	X
GIF	X	X
JPEG	X	X
JPEG2000	X	
PNG	X	X
Photoshop (.psd)	X	
TGA	X	
TIFF	X	X

Export Formats	Mac	Windows
OpenEXR	X	
WMPhoto		X

To export an image file:

1. Select **File > Export > Export Image File**.

   The Export Image File dialog box opens.

2. Select the export options, and then click **Render** or **Wireframe** to display the exported image preview.

Click to show/hide the parameters.

Parameter	Description
Export Area	
All Visible Objects	Exports an image that includes all visible objects (objects do not have to be currently on screen to be considered visible)
Current View	Exports an image that is exactly as it displays on the current screen
All Pages as Single Image	Saves all pages in the print area as one image. By default, the image dimensions are set to match the print area specifications in the Page Setup dialog box. The image dimensions can be changed, but must remain proportional to the print area aspect ratio.
Each Page as Separate Image	Saves each page in the print area as separate images. By default, the image dimensions are set to match the page size specifications in the Page Setup dialog box. The image dimensions can be changed, but must remain proportional to the page size aspect ratio.
Marquee	Exports an image within a user-created marquee. Select this option and then click **Draw Marquee** to temporarily close the dialog box. Click and drag to specify the area for export; the marquee dimensions are displayed on the Data bar. Click to set the export area and return to the Export Image File dialog box. The **Pixel Dimensions** of the image are automatically set to the marquee dimensions.
Dimensions	
Lock Aspect Ratio	Select to maintain the image aspect ratio when specifying dimensions
Resolution	Specifies the printed image resolution in pixels over inches
Pixel Dimensions	
Width/Height	Specifies the exported image dimensions; if **Lock Aspect Ratio** is selected, changes to one dimension will update the other to maintain the aspect ratio
Print Size	
Width/Height	Specifies the printed image dimensions in the selected **Unit**; if **Lock Aspect Ratio** is selected, changes to one dimension will update the other to maintain the aspect ratio
Units	Select a unit to apply to the **Print Size** parameters
Preview	Displays a rendered or wireframe preview according to the current settings
Render	Updates the preview with a rendered view using the currently set rendering option
Wireframe	Updates the preview with a wireframe view

Parameter	Description
Memory Required/ Estimated File Size	Estimates the amount of memory required for the export and the approximate file size of the exported file, based on the current settings
Update	Updates the estimated memory and file size requirements
Format	
File Type	Select the file format for the exported image
Color	Specify whether to export the image in Full color, Grayscale, or Black and white
Quality	For file formats that observe a quality setting, drag the **Opacity** slider to the left to decrease the image quality or to the right to increase quality
Update visible out of date viewports prior to exporting	Automatically updates any visible, out-of-date viewports before exporting
Reset all plug-in objects that require a reset prior to exporting	Automatically resets plug-in objects that require an update (such as data stamps) before exporting
Export Georeferencing File (Vectorworks Architect or Landmark required)	Available only if the current design layer is georeferenced. Along with the image file, exports a worldfile that describes the exported image's location, scale, and rotation in the geographic coordinate system. If multiple georeferenced layers are visible, the worldfile will be written according to the topmost layer. See "GIS and Georeferencing" on page 775 for more information.

3. Click **Save**.

   In the dialog box which opens, provide a name for the file; the program adds the file extension according to the file format selected. If the **Each Page as a Separate Image** was selected for export, the program automatically appends an incremental number to each file name.

4. Click **Save**.

   A new file is created without changing the original drawing file. This new file can then be opened in other applications or imported into another software program.

Importing and Exporting in Metafile Format

There are two versions of Metafiles (Windows only): standard (pre-Windows 95) and enhanced. The Vectorworks program supports the enhanced version.

Importing Metafile Format

Graphics files in the Metafile format can be imported from virtually any program, including AutoCAD and word processing programs.

When you import a Metafile file with the **Import Metafile** command, multiple bitmap objects may be created, which can be edited individually later on. When you import a Metafile file with the **Import Metafile as Bitmap** command, the image is imported as a single bitmap object. The **Trim**, **Clip**, and **Shear** tools cannot be used on the imported bitmap. Bitmap objects imported with these commands automatically have PNG compression applied.

To import a Metafile:

1. Select **File > Import > Import Metafile** or **File > Import > Import Metafile as Bitmap**.

   The Import EMF File dialog box opens.

2. Select the file to import, and click **Open**.

Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

Exporting Metafile Format

Graphics files can be exported in Metafile format for inclusion in virtually any Windows program, including AutoCAD and word processing programs.

To export a Metafile:

1. Select File > Export > **Export Metafile**.

   The Export Metafile dialog box opens.

2. Set the desired **Resolution** and click **OK**.

   In addition to rendered design layers and Vectorworks geometry, the **Resolution** setting affects PDF objects, which are rasterized when exported to Metafile format.

3. Specify the export file name and destination, and then click **Save**.

Importing and Exporting PDF

The Vectorworks Design Series products can import and export Portable Document Format (PDF) files. An imported PDF is useful for displaying company logos, contractor drawings, and other graphics.

In Vectorworks Design Series, the **Export PDF** and **Publish** commands create high-quality PDF files for printing, and takes advantage of PDF layers for creating an interactive representation of a model.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing PDF Files

The **Import PDF** command allows a PDF file to be imported into a Vectorworks file, where each PDF page becomes an independent PDF Page object. Create a reference to the original PDF file, if the original file may change, and you want to keep the imported pages up to date.

To import a PDF file:

1. Select **File > Import > Import PDF**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

2. Select the PDF file to be imported, and click **Open**.

   The Import PDF dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Import Range	Specifies which portion of the PDF file to import
Pages	Imports the specified range of pages from the PDF file
All Pages	Imports all pages in the PDF file
Reference	References the original PDF file (master file), so that the imported PDF pages can be updated when the master file changes. See “Workgroups and Referencing” on page 207 for more information.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Absolute path	Stores the absolute file path of the master file. Select this option when the location of the master file will not change in the future, or if the master file is on another volume.
Path relative to current document	Stores the file path of the master file relative to the target Vectorworks file; this option is available only if the master file is on the same volume as the Vectorworks file. Select this option if the target file and the master file may be moved to another volume in the future.
Save referenced cache to disk	Saves copies of the referenced PDF pages in the target Vectorworks file; if deselected, the target file is smaller, and the PDF pages are automatically updated when the target file is opened
Automatically update reference during file open	Updates the PDF pages from the master file each time this target file is opened, if the PDF pages are out of date

3. Click **Import**. Each PDF file page becomes a separate PDF page or referenced PDF page in the Vectorworks file.

   The PDF page can be resized, moved, or rotated with the **Modify > Rotate > Rotate** command or the **Rotate** tool, scaled with the **Modify > Scale Objects** command, printed, and exported. You can snap to geometry in the PDF page by selecting **Snap to Geometry** in the Object Info palette.

   A referenced PDF file can be opened with the default application associated with this file type. To open a referenced PDF file, use one of the following methods:

   - Ctrl-click (Mac) or right-click (Windows) the referenced PDF page in the drawing area, and then select **Open**.
   - Ctrl-click (Mac) or right-click (Windows) the referenced PDF file in the Navigation palette, and then select **Open**.
   - Select the referenced PDF file in the Navigation palette, and then select **Open** from the **Navigation** menu.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing and Exporting PDF
Snapping to and Scaling Imported PDF Files
Cropping Imported PDF Files

Snapping to and Scaling Imported PDF Files

Content files from a variety of manufacturers can be imported into a Vectorworks file in PDF format and then, because the Vectorworks program can snap to the vector graphics within the PDF Page object, the PDF object can be moved and snapped into a desired position. Lines drawn in the Vectorworks file snap to the 2D items in the PDF object, so the PDF can be incorporated into your own designs.

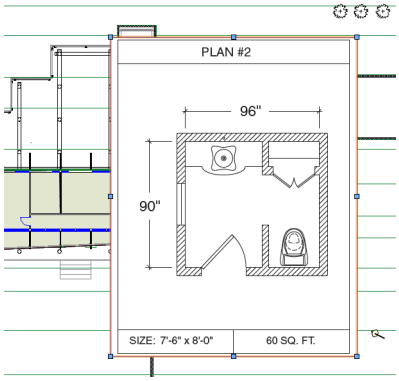
The **Scale Objects** command scales an imported PDF object so it matches the current drawing scale. Cropped PDF objects can also be scaled.

Snapping to a PDF file is only possible when the PDF originated from an application that creates vector graphics. Snapping cannot occur if the PDF file is encrypted. To enable snapping, select **Snap to Geometry** in the Object Info palette after importing.

To use an imported PDF file effectively:

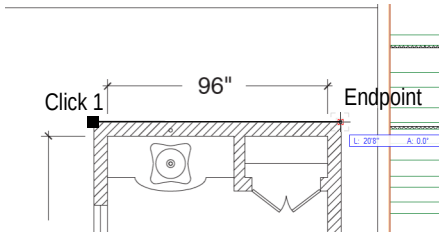
1. Import the PDF file as described in “Importing PDF Files” on page 1686.

   A PDF Page object is created. In this example, the PDF object imported at an incorrect scale for the current drawing.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

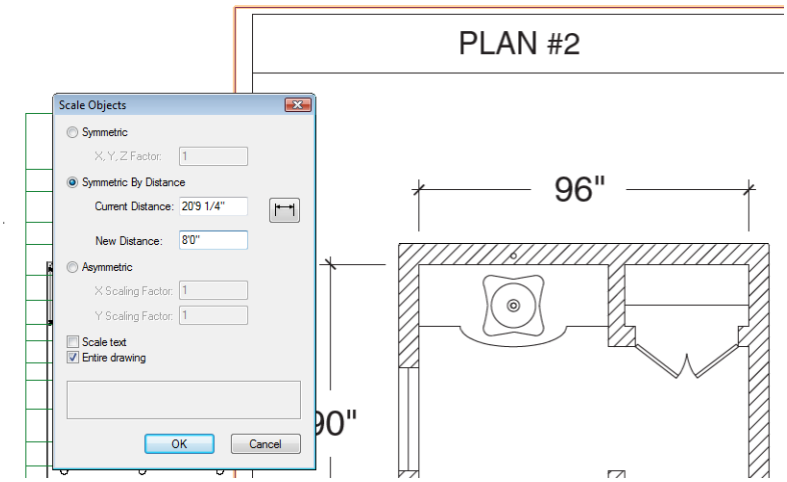


2. To change the scale of the imported PDF, select **Modify > Scale Objects**.

   The Scale Objects dialog box opens. The **Scale Objects** command is described in "Scaling Objects" on page 1063. Select **Symmetric by Distance**, and click the button to specify the current distance on the drawing. Click to indicate a segment of known distance in the PDF, and click again to finish the segment (note that snapping is possible within the PDF).

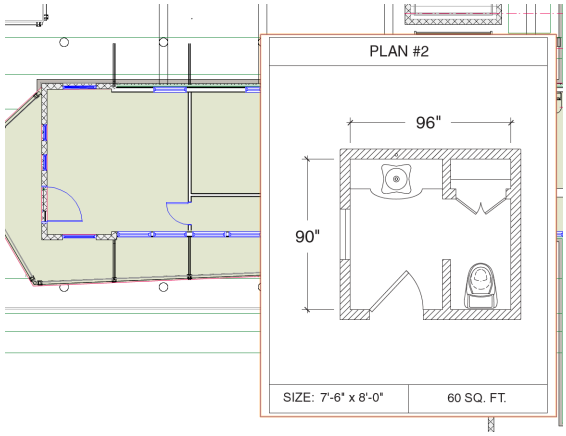


3. In the Scale Objects dialog box, enter the **New Distance** for the segment.



4. Click **OK**.

   The PDF object scale matches the drawing.



The imported PDF could be made into a viewport and cropped for use in the drawing. In this example, walls could be drawn in the Vectorworks file using the PDF object as a basis for the walls.

Alternatively, you can ungroup the PDF to create Vectorworks objects from the snapping objects, giving you a close approximation of the original PDF that can be manipulated in Vectorworks.

Importing PDF Files

Cropping Imported PDF Files

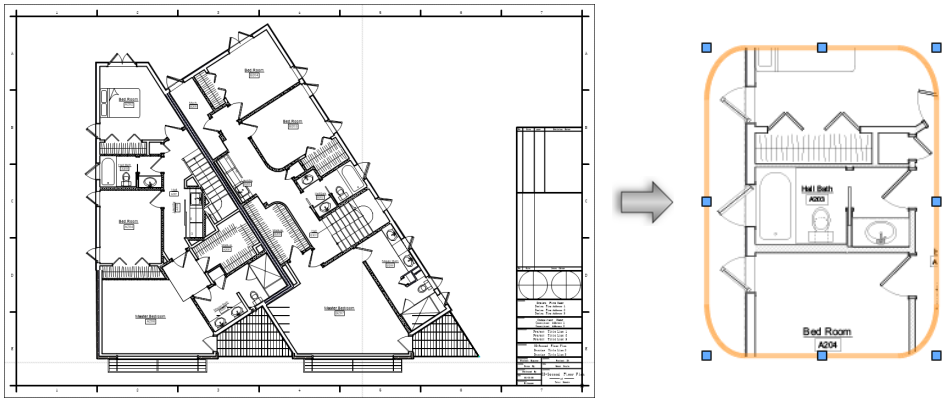
Imported and referenced PDF pages can be cropped so only part of the PDF page is visible.

To crop an imported of referenced PDF:

1. Import the PDF as described in “Importing PDF Files” on page 1686.
2. Select the PDF page object.
3. Select **Modify > Edit Crop**.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a PDF and select **Edit Crop** from the context menu, or simply double-click on the PDF page object.
4. A colored border around the PDF indicates you are in an editing mode. The **Exit PDF Crop** command becomes available from the **Modify** menu, and the **Exit PDF Page Crop** button is visible in the top right corner of the drawing window. If the PDF object is on a 3D plane, the view will rotate so that the view is parallel to the screen plane
5. Create a 2D object such as a rectangle, circle, or polyline. The 2D object must define an area; for example, a 2D line cannot be used. Position the 2D object to delimit the new PDF display area. The fill of a PDF cropping object is always None; however, the pen style can be set from the Attributes palette while in Edit Crop mode. Move and resize the 2D object as needed.
6. Click **Exit PDF Page Crop** to return to the drawing.
7. The cropped PDF displays; in the Object Info Palette, the **Crop** status changes to Yes.
8. To change, replace, or delete the crop object, select the PDF page object and then select **Modify > Edit Crop** to re-enter Edit Crop mode.

   To change the visibility of the crop object, change the **Crop Visible** setting in the Object Info palette.



Importing PDF Files

Exporting PDF Files

Vectorworks drawing files can be exported to PDF for use as an interactive representation of a model, or for final quality printing purposes. Export settings are designed for maximum usability and the smallest possible file size. Pages in the Vectorworks file become PDF pages when exported. Fonts are always embedded in the PDF, for portability. Layer transfer modes other than **Paint** become transparent in the PDF. The Vectorworks program can export the currently visible design layer(s) or sheet layer, or create a batch export list of saved views and sheet layers.

You can choose between the standard PDF format and PDF/A-1b. The PDF/A-1b format is useful for archiving, and helps preserve the visual appearance of the document by embedding color and font information into the exported file. PDF/1-1b format flattens all layers, and elements of the drawing with transparency settings appear opaque.

The layers and classes in the Vectorworks file can be exported as PDF layers, to create an interactive model representation (PDF layers require PDF rev. 1.5 minimum).

You can export a single PDF file, or select the **Publish** command to export several files as a batch.

Batch PDF File Export

Use the **Publish** command to export a series of sheet layers and/or saved views from the current drawing and externally-referenced files to PDF. See “Batch Publishing” on page 1765.

Single PDF File Export

The **Export PDF** command creates a PDF file from the current sheet layer or visible design layer(s).

To export the current sheet layer or design layer(s) to PDF:

1. Set the view for the export. The current sheet layer or currently viewed design layer(s) are exported.

   Set the visibility of layers and classes as desired. All visible and grayed layers are exported; invisible layers and classes are not exported. For viewports on sheet layers, set the layer and class visibility of each viewport.

2. Select **File > Export > Export PDF**.

   The Export PDF dialog box opens.

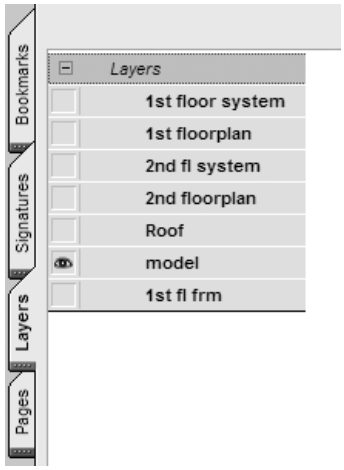
3. Specify the settings for exporting to PDF.

Click to show/hide the parameters.

Parameter	Description
PDF Conversion	
Export Design Layers as PDF Layers (PDF format only)	When exporting design layers, creates a PDF layer from each visible design layer
Export Classes as PDF Layers (PDF format only)	When exporting design layers, creates a PDF layer from each visible class
Make grayed PDF Layers initially invisible (PDF format only)	When creating PDF layers, grayed layers or classes appear invisible when first opening the resulting PDF
but appear with normal attributes when made visible	When creating PDF layers, grayed layers or classes do not display grayed, but with their assigned attributes once made visible in the PDF
Format	Select the PDF export format
Resolution	Sets the export resolution of pictures, rendered images, and patterns (unless **Export patterns at on-screen resolution** is selected). A larger value results in higher quality images, but also a larger PDF file. *Range*: 72—2500 dpi.
Export patterns at on-screen resolution	Exports the pattern size as viewed on-screen; when deselected, patterns are exported according to the specified **Resolution**
Downsample higher resolution raster images to	Reduces the resolution of raster images to the specified value, resulting in a smaller PDF file size
Rasterize text	Exports text in bitmapped format to ensure that it matches the on-screen appearance; use this option when the font is not displaying correctly in a bold or italic style. This option may result in a larger PDF file and can cause the font to appear pixellated at high PDF zoom levels.
Use Opacity of	Sets the transparency of layers in the PDF file (replaces Layer Transfer modes in the Vectorworks program). Not necessary on Mac systems or on Windows systems when the GDI+ imaging Vectorworks preference is enabled.
Gray level for grayed Layers and Classes	Adjusts the level of gray when exporting with grayed layers and/or classes
Open PDF in the default viewer	After export, opens the PDF automatically in the system's specified PDF viewer
Export Range	
Export the whole printable area as one page	When the printable area consists of more than one page, exports the entire area as a single PDF page
All Pages	Exports all pages, creating a PDF page for each Vectorworks page
Pages	Exports the specified pages only
Current View	Exports the current view to a PDF page
Update visible out of date viewports prior to exporting	Automatically updates any visible, out-of-date viewports before exporting
Reset all plug-in objects that require a reset prior to exporting	Automatically resets plug-in objects that require an update (such as data stamps) before exporting
Recalculate worksheets prior to exporting	Automatically recalculates worksheets before exporting

4. Click **OK** to create the PDF file based on the settings. If **Open PDF in the default viewer** was selected, the PDF viewer opens automatically.

   If layers or classes were exported as PDF layers and the viewer supports PDF layers, the layers are listed and their visibility can be controlled through the viewer.



AL Importing and Exporting in Shapefile Format

Importing in Shapefile Format

The Vectorworks Architect and Landmark products can import files in shapefile (.shp) format. Each geometric feature stored in the shapefile is converted to a Vectorworks object. In turn, each of these objects has a corresponding data record, which is converted from the shapefile's database file.

To import a shapefile and select conversion options:

1. Select **File > Import > Import Shapefile**, and then select the shapefile to import.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

2. The Vectorworks program determines the appropriate units for the imported objects from the projection (.prj) file, if one is available. If no projection file is found, the current document's units are used.
3. The Import Shapefile dialog box opens. Specify the file import options.

Click to show/hide the parameters.

Parameter	Description
Import to layer	Select the design layer to import themes from the shapefile into; by default, objects will be imported to a new layer named after the shapefile If a projection file (.prj) is included with the shapefile, the layer is automatically set to be georeferenced based on the projection file (but it can be changed if necessary)
Edit Layer	Opens the Edit Design Layers dialog box, to edit the selected design layer's georeferencing settings as needed
Georeference message	If the selected layer is georeferenced, a message displays the current projection setting; otherwise, a message indicates that the layer is not georeferenced
Assign to class	Select an existing class to assign to the themes imported from the shapefile, or select New to create a new class

4. Click **OK** to import the file with the selected options. The Object Info palette displays the imported shapefile information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Exporting in Shapefile Format
GIS and Georeferencing

Exporting in Shapefile Format

The **Export Shapefile** command exports drawing components to the shapefile format. Each shapefile (.shp) exported is accompanied by an index file (.shx) and a database file (.dbf) with the same name.

Shapefile export options allow the export of drawing components using multiple criteria. For an object to be exported, it must satisfy all selected criteria. For example, if you select both **Objects with classes** and **Objects in layers** export options, only objects that belong to both one of the selected classes and one of the selected layers will be exported.

The following objects cannot be exported: text, dimensions, worksheets, bitmaps, lights, layer links, walls, round walls, and some 3D objects (including extrudes, sweeps, meshes, roofs, floors, columns, and NURBS curves). Other objects are exported as follows:

- Lines, rectangles, circles, ovals, arcs, polygons, rounded rectangles, polylines, and path plug-in objects are exported as shapefile polygons or polylines.
- Symbols, 2D and 3D loci, and point plug-in objects are exported as shapefile points or 3D points.
- 3D polygons are exported as shapefile 3D polyline or 3D polygon shape types.
- Shapefiles can contain only one shape type. If multiple shape types are being exported, they will go into separate files.
- Groups that contain exportable object types are exported as individual polygons, polylines, and loci, as appropriate.
- If the Vectorworks file contains projection data, the file can either be deprojected, or exported with projection and (optionally) a projection (.prj) file.
- Object data that are not part of the record can be included in the export.

To export a shapefile:

1. Select **File > Export > Export Shapefile**.
2. The Export Shapefile dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Objects with record	Select which objects that share a specific data record will be exported
Objects with classes	Select this option to export objects in specified classes
Objects in layers	Select this option to export objects in specified design layers
Selected objects	Select this option to export objects that are currently selected
Deproject / Keep projection	Select **Deproject** to create a shapefile with latitude and longitude coordinates only; select **Keep Projection** to create a shapefile with the currently projected coordinates (in document units)
Write projection files	If **Keep projection** is selected, select this option to create a projection file (.prj), in addition to the .shp file
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Additional Object Data	Select this option to export additional object data that belong to the object(s) being exported, but are not part of the record

3. Select the export parameters.

   If **Objects with classes** or **Objects in layers** is selected, click **Select Classes/Select Layers** to open the relevant dialog box. Place a check mark in the column beside all classes and layers from which to export objects, and click **OK**.

   If **Additional Object Data** is selected, the Additional Object Data dialog box opens automatically. Place a check mark in the column beside additional data to be exported. If desired, rename data in the **Export Name** field; names can be no more than ten characters long. Click **OK**. The customizations in the list are remembered across Vectorworks sessions.

   Click **OK**.

4. Specify the file name and destination, and then click **Save**. The default shapefile file name is the record format name.

   The Vectorworks program creates a set of export files, leaving the original file intact.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing in Shapefile Format
GIS and Georeferencing

Importing and Exporting Scripts

Exporting a Simple VectorScript (3D only)

You can export 3D drawing objects as a Pascal-like script designed to be easy to import into programs like Strata Software products.

To export a simple VectorScript file:

1. Select **File > Export > Export Simple VectorScript (3D only)**.
2. Select whether to export in front view or in the current view, and click **OK**.
3. Specify the export file name and destination, and then click **Save**.

Importing Scripts

The **Import Script** command executes a saved script (Python or VectorScript), automatically creating a new design layer for the scripted objects. See “Using Scripts” on page 1771.

To import a script:

1. Select **File > Import > Import Script**.
2. Select a file from the Open dialog box.
3. Click **Open**.

Exporting Scripts

The **Export Script** command writes out the contents of the current file as a script. The script can then be imported into another file, used as part of another script, or studied as a guide for learning. See “Using Scripts” on page 1771.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To export a script:

1. Select **File > Export > Export Script**.
2. Specify the export file name and destination, and then click **Save**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Scripts

Importing and Exporting in 3ds Format

The Vectorworks Design Series products can both import from and export to the .3ds file format used by Autodesk® 3ds Max® software, version 3.0 and higher.

Importing in 3ds Format (3D only)

When importing a 3ds file, specify how the drawing components will be represented in the Vectorworks file.

To import a 3ds file:

1. Select **File > Import > Import 3DS (3D only)**, and then select the 3ds file to import.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.
2. The Import 3DS Options dialog box opens; the options available depend on the contents of the file being imported. Specify the file import options.

Click to show/hide the parameters.

Parameter	Description
Import Materials	Select whether to assign fill colors from the imported objects according to the colors of the materials of objects when selected.
Import Transparency and Image Data as Renderworks Textures	Select to import texture and transparency materials as Renderworks texture resources
Create Renderworks Textures for All Materials	Select to create Renderworks textures for all imported materials from the input file
Import Mesh Objects	Imports each mesh object defined in the 3ds file as either a group of mesh objects or 3D polygons
Import as Vectorworks Mesh Object	Select to import all geometry as a group of mesh objects The 3DS file format does not contain smoothing information. The 3DS import will set document smoothing to the created mesh objects.
Import as Groups of 3D Polygons	Select to represent all imported geometry as a group of 3D polygons
Preserve Texture Assignment and Mapping	If **Import Materials** is selected, and you are importing geometry as polygons, specify whether textures and texture mappings are applied to imported 3D polygons to preserve their overall appearance (Renderworks required).
Scale	Opens the Import 3DS Scale dialog box, for specifying a scale to apply to all imported objects; see “Scaling 3ds Objects” on page 1697
Import Light Objects	Imports light objects from the 3ds file
Import Spot Light	Select this option to import spot lights from the 3ds file as Vectorworks spot lights
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Import Omni Light	Select this option to import omni lights from the 3ds file as Vectorworks point lights
Import Camera Objects	Imports camera objects from the 3ds file
Import Active Camera Only	Imports only the active camera from the 3ds file, and sets the current view to the camera view
Create Saved Views for Each Camera	Imports all camera objects from the 3ds file, and creates a saved view in the Vectorworks file for each

3. Click **OK** to import the file. The Object Info palette displays the image information.

Exporting in 3ds Format (3D only)
Scaling 3ds Objects

Exporting in 3ds Format (3D only)

Different portions of the drawing can be selected for export to a 3ds file.

To export a 3ds file:

1. If 3D objects from only one layer should be exported, make that layer active. If only selected objects should be exported, select the objects.
2. Select **File > Export > Export 3DS (3D only)**.
3. The Export 3DS Options dialog box opens; the options available on the lower part of the dialog box depend on which group of items were selected for export.

Click to show/hide the parameters.

Parameter	Description
Export	
Selected Objects on Active Layer	Exports only selected 3D objects from the active layer; if no objects are selected, all 3D objects from the active layer are exported
All Visible Objects on Active Layer	Exports all visible 3D objects from the active layer (including objects in design layer viewports). Exclude or include specific types of objects using the dialog check boxes; for example, export walls but not lights.
All Visible Objects on All Layers	Exports all visible 3D objects from all layers in the Vectorworks file (including objects in design layer viewports). Exclude or include specific types of objects using the dialog check boxes; for example, export walls but not lights.
Export Quality (Number of Polygons)	Controls the conversion quality of 3D objects that have rounded sides. Move the slider bar left to convert rounded objects using fewer large polygons (thus lower quality). Move the bar right to create rounded objects of higher quality. Exporting more polygons slows down the export and increases the size of the output file.
Export Objects	Exports all kinds of Vectorworks 3D objects except walls, 3D loci, symbols, lights, and plug-in objects
Export 3D Locuses	Exports each 3D locus as a one-vertex 3ds mesh
Export Walls	Exports walls

Parameter	Description
Export 3D Symbols	Exports each 3D symbol as a group of 3D meshes
Export Plug-in Objects	Exports each plug-in object as a group of 3D meshes
Export Lights	Exports Vectorworks light objects as 3ds light objects. Vectorworks point lights are exported as 3ds omni lights, and Vectorworks directional and spot lights are exported as 3ds spot lights.
Export Current View as Camera	If the current view was set with one of the perspective options (Narrow Perspective, Normal Perspective, Wide Perspective, or a custom perspective), this option exports the view as a 3ds camera object
Scale	Opens the Export 3DS Scale dialog box, for specifying a scale to apply to all exported objects; see “Scaling 3ds Objects” on page 1697
Export Materials	Exports colors that are attached to the exported objects as 3ds materials; this includes colors both from the Attributes palette and from Renderworks textures
Export Textures as	If **Export Materials** is selected, this option exports each Renderworks texture to a texture image file (Renderworks required). Select the export file format from the list. Exported image files are automatically named with sequential numbers starting at zero. For example, a Vectorworks file with two textures would export two image files named Textr0.jpg and Textr1.jpg. When exporting multiple drawings with textures, be sure to save them to different folders—otherwise, new textures will overwrite existing exported textures.

4. Select the types of objects to export, and then click **OK**.
5. Specify the export file name and destination, and then click **Save**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing in 3ds Format (3D only)
Scaling 3ds Objects

Scaling 3ds Objects

The scale of 3ds objects can be changed during both import and export. The scaling is applied to all imported/exported objects.

To scale 3ds objects during import or export:

1. From the import or export dialog box, click **Scale**.
2. The Import/Export 3DS Scale dialog box opens. Choose whether to scale the object(s) by custom dimensions or by scale factor.

Click to show/hide the parameters.

Parameter	Description
(3DS) Model Width/Height/Depth	Displays the original dimensions of the object(s) selected for import or export; if there are multiple objects, the scaling center is the center of the selection bounding box
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Custom	Allows entry of custom dimensions for imported or exported objects in the **Desired Width**, **Desired Height**, and **Desired Depth** fields. Select **Maintain Proportions** to have the Vectorworks program automatically adjust the proportions when a custom width, height, or depth is entered.
Scale	Allows entry of a scaling factor to be applied to imported or exported objects. Either select a **Scale** from the list, or select Custom from the top of the list and enter a **Custom Scale** value. When the scale factor changes, the dimension values also change to show the exact size of the selection bounding box for the given scale.

3. Click **OK** to save the scaling information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing in 3ds Format (3D only)
Exporting in 3ds Format (3D only)

Importing and Exporting in IGES Format

Importing IGES Format (3D only)

During IGES import, points are imported as a group of 3D loci, NURBS curves are imported as a group of NURBS curves, NURBS surfaces are imported as a group of NURBS surfaces, closed solids are imported as a group of imported solids, and open solids are imported as a group of NURBS surfaces. If there is only one element in the IGES file, it is imported as a single element rather than grouped. Imported solids cannot be ungrouped or edited through the **Edit Group** command; they can be used in solid operations.

To import an IGES file:

1. Select **File > Import > Import IGES**.

   The Import IGES File dialog box opens.

2. Select the file to import, and click **Open**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

Exporting in IGES Format (3D only)

The **Export IGES (3D only)** command exports 3D curves, surfaces, and solids from a Vectorworks file into IGES format (version 5.3). Closed solids are exported as solids, while open solids are exported as a collection of surfaces.

Objects that are exported to an .igs file include:

- 3D loci
- Chamfers
- Fillets
- Parametric objects
- Sweeps
- 3D polygons
- NURBS curves and surfaces
- Groups
- Shells
- 3D Symbols
- Architectural objects (wall/slab/column/stair)
- CSG solids (add/subtract/intersect/section)
- Extrudes and multiple extrudes
- Solids (sphere/cone/hemisphere)

To export an IGES file:

1. Select **File > Export > Export IGES (3D only)**.

   The Export Options dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

2. Select **Export Solid as Trimmed Surfaces** to export a solid as several different "bodies" (for example, a cube exports as six bodies); otherwise, a solid is exported as a single body.
3. Click **OK**.
4. Specify the export file name and destination, and then click **Save**.

Export Item	Export Result
NURBS curve	IGES Entity Type 126
NURBS surface	IGES Entity Type 128
Trimmed NURBS surface	IGES Entity Type 144
Closed solid	IGES Entity Type 186

Importing and Exporting in SAT Format

Importing SAT (3D only)

ACIS/SAT 3D solids are imported as NURBS-based Vectorworks solids.

To import an SAT file:

1. Select **File > Import > Import SAT**.

   The Import SAT File dialog box opens.
2. Select the file to import, and click **Open**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

Exporting SAT (3D only)

Vectorworks software creates an SAT file for exporting ACIS 3D solids.

To export an SAT file:

1. Select **File > Export > Export SAT (3D only)**.

   The Export Options dialog box opens.
2. Select **Export Solids as Trimmed Surfaces** to export a solid as several different ACIS "bodies" (for example, a cube exports as six ACIS bodies). If this option is deselected, a solid is exported as a single body.
3. Click **OK**.
4. Specify the file name and destination, and then click **Save**.

Importing and Exporting in STEP Format

Importing STEP Format (3D only)

During STEP import, points are imported as a group of 3D loci, NURBS curves are imported as a group of NURBS curves, NURBS surfaces are imported as a group of NURBS surfaces, closed solids are imported as a group of imported solids, and open solids are imported as a group of NURBS surfaces. If there is only one element in the STEP .stp file, it is imported as a single element rather than grouped. Imported solids cannot be ungrouped or edited through the **Edit Group** command; they can be used in solid operations.

To import a STEP file:

1. Select **Files > Import > Import STEP (3D only)**.

The Import STEP File dialog box opens.

2. Select the file to import, and click **Open**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

Exporting STEP (3D only)

The **Export STEP (3D only)** command exports 3D curves, surfaces, and solids from a Vectorworks files into STEP format. Closed solids export as solids, while open solids export as a collection of surfaces.

Objects that are exported to an .stp file include:

- 3D loci
- Chamfers
- Fillets
- Parametric objects
- Sweeps
- 3D polygons
- NURBS curves and surfaces
- Groups
- Shells
- 3D Symbols
- Architectural objects (wall/slab/column/stair)
- CSG solids (add/subtract/intersect/section)
- Extrudes and multiple extrudes
- Solids (sphere/cone/hemisphere)

To export a STEP file:

1. Select **File > Export > Export STEP (3D only)**.

   The Export Options dialog box opens.

2. Select **Export Solid as Trimmed Surfaces** to export a solid as several different "bodies" (for example, a cube exports as six bodies); otherwise, a solid is exported as a single body.
3. Click **OK**.
4. Specify the file name and destination, and then click **Save**.

Importing and Exporting in Rhino 3DM Format

Importing Rhino 3DM Format (3D only)

During Rhino 3DM import, points are imported as a group of 3D loci, NURBS curves are imported as a group of NURBS curves, NURBS surfaces are imported as a group of NURBS surfaces, closed solids are imported as a group of imported solids, and open solids are imported as a group of NURBS surfaces. If there is only one element in the Rhino 3DM file, it is imported as a single element rather than grouped. Imported solids cannot be ungrouped or edited through the **Edit Group** command; they can be used in solid operations.

Exporting in Rhino 3DM Format (3D only)

The **Export Rhino 3DM (3D only)** command exports 3D curves, surfaces, and solids from a Vectorworks file into Rhino 3DM format, used by Rhinoceros® model files. Closed solids are exported as solids, while open solids are exported as a collection of surfaces.

Objects that are exported to a .3dm file include:

- 3D loci
- Chamfers
- Fillets
- 3D polygons
- NURBS curves and surfaces
- Groups
- Architectural objects (wall/slab/column/stair)
- CSG solids (add/subtract/intersect/section)
- Extrudes and multiple extrudes

- Parametric objects
- Sweeps
- Shells
- 3D Symbols
- Solids (sphere/cone/hemisphere)

To export a Rhino 3DM file:

1. Select **File > Export > Export Rhino 3DM (3D only)**.

   The Export Options dialog box opens.
2. Select **Export Solid as Trimmed Surfaces** to export a solid as several different "bodies" (for example, a cube exports as six bodies); otherwise, a solid is exported as a single body.
3. Click **OK**.
4. Specify the file name and destination, and then click **Save**.

Importing and Exporting in Parasolid X_T Format

Importing Parasolid X_T Format (3D only)

The Vectorworks program can import 3D Parasolid X_T objects, including NURBS curves, NURBS surfaces, and solids; solids are imported as generic solids.

To import a Parasolid X_T file:

1. Select **File > Import > Import Parasolid X_T**.

   The Import Parasolid X_T File dialog box opens.
2. Select the file to import, and click **Open**.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

Exporting in Parasolid X_T Format (3D only)

The **Export Parasolid X_T (3D only)** command exports 3D objects to Parasolid .x_t format. Only 3D objects are exported. This is a convenient way of exchanging files with other applications that also use Parasolid as their modeling kernel, even when the other application's Parasolid version is older.

To export a Parasolid X_T file:

1. If 3D objects from only one layer should be exported, make that layer active. If only selected objects should be exported, select the objects.
2. Select **File > Export > Export Parasolid X_T (3D only)**.

   The Export Parasolid X-T Options dialog box opens. Select whether to export only selected objects from the active layer, all visible objects from the active layer, or all visible objects from all layers in the file.

Click to show/hide the parameters.

Parameter	Description
Export	
Selected Objects on Active Layer	Exports only selected 3D objects from the active layer
All Visible Objects on Active Layer	Exports all visible 3D objects from the active layer
All Visible Objects on All Layers	Exports all visible 3D objects from all layers in the Vectorworks file

Parameter	Description
Parasolid Version	Specify the Parasolid version for export; if the exported file cannot be opened, try exporting to a lower version of Parasolid

3. Click **OK**.
4. Specify the export file name and destination, and then click **Save**.

Importing and Exporting Georeferenced Raster Images

A georeferenced raster image is a regular image file that comes with an additional companion file, known as a world file. A world file associated with the raster file must have the same name as the raster file and an extension of .bpw, .jgw, .pgw, .tfw, .gfw, or .wld. Georeferenced raster images are supported in the Vectorworks Landmark product, for both import and export.

Importing Georeferenced Raster Images

When you import a raster image file, the Vectorworks Landmark product automatically uses the companion world file to import the image as a georeferenced raster image.

To import a georeferenced raster image:

1. Select **File > Document Settings > Units** and set the document units to match the units of the image to be imported. For example, if the world file's units are meters, set the Vectorworks document units to meters.
2. Select **File > Import > Import Image File.**

   The Open dialog box opens. Select the raster file for import.
3. If a world file is associated with the raster file, the image is automatically imported, translated, and scaled. The current view changes to Fit to Objects, so the imported image is visible.

   To import an image without its associated world file scaling and translation, either rename the associated world file, or move it to a different folder.

   GeoTiff image files can be imported, but no georeferencing information within the file is used.

Exporting Georeferenced Raster Images

If the current design layer is georeferenced, there is an option to export a worldfile along with an image file of the Vectorworks drawing.

To export a georeferenced raster image and world file:

1. Select **File > Export > Export Image File.**
2. The Export Image File dialog box opens. Select the export options, including **Export Georeferencing File**.
3. Click **Save**, and then name and save the image file. Along with the image file, the Vectorworks program exports a worldfile that describes the exported image's location, scale, and rotation in the geographic coordinate system. If multiple georeferenced layers are visible, the worldfile will be written according to the topmost layer. See "Exporting an Image File" on page 1683.

Importing and Exporting Vectorworks Spotlight Data

The Vectorworks Spotlight product provides the ability to import and export lighting design data. The Vectorworks program is compatible with Lightwright 5 and later, and includes extra functionality to automatically data transfer to and from Lightwright.

Exporting Instrument Data

Vectorworks Spotlight instrument data (including accessory, power, and position information) can be exported to an external file. The resulting file can then be opened by a spreadsheet or database program, and is compatible with Lightwright.

All instrument and accessory data from the current light plot are exported. Individual instruments cannot be selected for export.

To export instrument data:

1. Select **File > Export > Export Instrument Data**.

   The Export Instrument Data dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Instrument Fields	Selects the instrument data to be exported. Double-click (Macintosh) or click (Windows) an item to select it; a plus sign (+) next to the field indicates that it will be exported. To deselect a field, double-click it.
All	Selects all instrument data fields for export
Lightwright	Selects only the instrument data fields supported by Lightwright. All Lightwright-compatible fields will be exported; these fields are displayed with a plus sign. Verify the selections to ensure that these are the fields desired for export.
User	Restores the last saved or loaded set of instrument data fields
None	Deselects all instrument data fields
Save	Saves the current set of selections; enter a name for the selection set file
Load	Loads a selection set file
Export field names as first record	Retains headers for the field information during export; this option is recommended, as it helps to organize the data in the database or spreadsheet program
Export in Lightwright 4 format	Exports all static accessories attached to an instrument as a list separated by a “+” sign, to the Accessories field associated with the instrument

The Unique ID field (UID) is not in the list, but is automatically exported. This field is required to re-import data.

2. Select the desired parameters and click **Export**.
3. Specify the export file name and destination, and then click **Save**.

4. The selected instrument data is exported to the file in tab-delimited format. Import the file as tab-delimited into the other program.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing and Exporting Vectorworks Spotlight Data
Automated Import and Export to Lightwright

Exporting Patch Information

The **Export ASCII Patch** command exports the dimmer and channel information from the drawing to a USITT standard ASCII text patch file (Version 3.0, Ident 3.0).

To export dimmer and channel information to a patch file:

1. Select **File > Export > Export ASCII Patch**.
2. Select whether to clear existing console patch information when importing the patch data into the lighting console. Click **Yes** to delete all existing patch data in the console before importing the new patch. Click **No** to preserve the current console patch data; the new patch information may overwrite existing data.
3. The Export ASCII Patch File dialog box opens. Specify the location and file name of the patch text file.

   Fixtures with multiple channels are patched sequentially. For example, setting the **Channel** to 6, the **Dimmer** to 20, and the **Num Channel** to 4 generates the following patch information:

Channel	Dimmer
6	20
7	21
8	22
9	23

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing and Exporting Vectorworks Spotlight Data

Importing Instrument Data

Instrument data from an external file can be imported into a Vectorworks Spotlight file. Import settings can be saved.

To import instrument data:

1. Select **File > Import > Import Instrument Data**.

   The Instrument Data Import dialog box opens.
2. Click **Browse** to search for the file to be imported.

   The imported file must consist of tab-delimited text, with the instrument information in the rows and the field information in the columns.
3. Once the file has been selected, choose the **Unique ID Field** from the list of available fields.

   This field will be used as the key field to match imported data to the data in the plot. Normally, select UID (Unique ID) or EID (External ID).
4. After the **Unique ID Field** has been selected, the list of incoming data fields is displayed in the **Incoming Data** list on the left, and all available instrument fields are shown in the **Record Fields** list on the right.

5. To map the incoming data to an instrument field, select a data item from the **Incoming Data** list, and then select the destination record field on the right. Click **Link** to link the fields. The linked field moves to the **Record Fields** list. (To unlink two linked fields, select the field from the **Record Fields** list and then click **Unlink**.)
6. Continue mapping fields individually until the desired links have been made. To save time, click **AutoLink**; this automatically matches fields based on the similarity of their names. Verify that the links were made as intended.
7. To import the data without saving the links, click **Import**.

   Saving the mapping information allows you to skip the process of mapping individual fields when importing similar external files. Once the fields have been mapped appropriately, save the mapping links by clicking **Save**. Specify the name of the links file, which can then be restored with the **Load** button.
8. During import, the Vectorworks Spotlight product attempts to match all incoming data to an appropriate instrument. However, if items exist which cannot be matched, these display as orphan instruments at the bottom left of the drawing, outside the page boundary.

   The first row of the imported file should contain the field labels. However, if the file to import contains instrument data in the first row, select **Import First Record**.

To load saved mapping files:

1. Click **Browse** and select the file to be imported.
2. Click on the **Load** button, and select either a pre-defined mapping file or a saved mapping file. Click **Import** to open the mapping file and automatically map the fields based on the saved links.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing and Exporting Vectorworks Spotlight Data
Automated Import and Export to Lightwright

Automated Import and Export to Lightwright

The lighting instruments and parameters specified in the Vectorworks Spotlight product can be automatically sent to Lightwright versions 5 and later to perform paperwork processing. Any changes to the data in Lightwright automatically update the light plot. Once configured (and once the Vectorworks file has been saved), these updates occur seamlessly in both applications.

For versions of Lightwright earlier than 5, see “Exchanging Data with Older Versions of Lightwright” on page 1707.

To specify automatic data transfer between the Vectorworks Spotlight product and Lightwright:

1. Select **File > Document Settings > Spotlight Preferences**.

   The Spotlight Preferences dialog box opens. Click the Lightwright tab.

Click to show/hide the parameters.

Parameter	Description
Use automatic Lightwright Data Exchange	Enables the automatic exchange of data between the Vectorworks Spotlight product and Lightwright; deselect to suspend or stop the data exchange
Perform a complete export on exit	Exports the Spotlight data, replacing all the data in Lightwright the next time Lightwright is used. This should occur whenever field export changes have occurred; therefore, when changes have been made, the field appears dimmed and cannot be deselected.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Include Inventory	Select to include inventory data in the data transfer. An updated instrument types inventory list, which is controlled by the **Lighting Inventory Setup** command, is sent to Lightwright when the Lightwright data exchange performs a complete export and when lighting inventory is updated (see "Lighting Inventory Setup" on page 960 and "Inventory Reports" on page 969). Only instruments types with a count of one or more are sent to Lightwright. Inventory updates from Lightwright are processed with the other instrument data. Instrument summaries are updated automatically if **Show Counts** and **Compare to Inventory** are selected in the Instrument Summary Settings dialog box for the placed objects.
File Path	Specifies the location of the exchange file. The file is in .xml format, and it is named according to the Vectorworks file.
Same location as the file	Saves the exchange file with the Vectorworks file
A custom location	Saves the exchange file in a specific location; click **Choose** to set the location
Available Fields	Lists field names from the Vectorworks file that are available to export to Lightwright, but have not been selected for export
Move >	Moves the selected Vectorworks file field to the **Export Fields** list for export to Lightwright
< Move	Removes field names from the **Export Fields** list, returning them to the **Available Fields** list, where they are not included for export
Export Fields	Lists field names that will be exported to Lightwright
Reset Defaults	Restores the fields to the lists specified as a default when **Save as default** was last clicked
Save as default	Saves the current lists as the default mapping to be restored by **Reset Defaults**; the default settings saved include those on the Lighting Device and Universe tab

2. Click **Use automatic Lightwright Data Exchange** to enable automatic data transfer between the Vectorworks Spotlight product and Lightwright, and indicate the location for saving the transfer file.
3. Move fields to be exported from the **Available Fields** list to the **Export Fields** list. All parameters in the **Export Fields** list are exported to Lightwright.

   Click **Save as default** to save the current status of the lists for new Vectorworks files. When enabling data exchange, click **Reset Defaults** to use the default lists.
4. Click **OK** to close the dialog box and export the instrument and accessory information from the Vectorworks file to the exchange file.

As instruments are added, deleted, and changed in the Vectorworks Spotlight product, the edits appear in Lightwright. Similarly, edits made in Lightwright apply automatically to the Vectorworks file. Instrument data are updated when the file focus switches from the Vectorworks file to Lightwright, or Lightwright to the Vectorworks file.

Lights added in Lightwright are placed between existing lights on a lighting position upon import, based on the unit number of the new light. If no adjacent lights can be determined, the light is placed at the center of the lighting position.

In the Vectorworks file, an import worksheet is automatically created, listing instruments that have been added from Lightwright as well as instruments that did not have a Lightwright equivalent and should possibly be deleted from the Vectorworks file.

Right-click (Windows) or Ctrl-click (Macintosh) on a database row of the import worksheet, and choose **Select Item** to select and display the associated instrument.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Lighting Device Setup
Universe Assignment in Vectorworks Spotlight
Importing and Exporting Vectorworks Spotlight Data

Exchanging Data with Older Versions of Lightwright

The Vectorworks Spotlight product automatically exchanges data with Lightwright version 5 and later (see “Automated Import and Export to Lightwright” on page 1705). However, instrument data can also be transferred between the Vectorworks Spotlight product and versions of Lightwright earlier than 5. Once lighting instruments and parameters have been specified in the Vectorworks file, their information can be sent to Lightwright to perform paperwork processing. When the paperwork in Lightwright is complete, the data can be imported back into the Vectorworks file to update the light plot.

The Vectorworks Spotlight product includes Lightwright automated action files, which, when selected at installation, are located in the Extras folder. These files have the correct settings required for exchanging data with the Vectorworks Spotlight product, and need to be placed into Lightwright’s Automated Actions folder. Consult the Lightwright documentation for information on using these files.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Exporting the Vectorworks Data for Editing
Importing the Vectorworks Data into Lightwright
Exporting the Edited Data from Lightwright
Importing the Edited Lightwright Data into Vectorworks

Exporting the Vectorworks Data for Editing

To export instrument data from the Vectorworks Spotlight file to Lightwright:

1. When the Vectorworks file contains instrument data that is ready to export, select **File > Export > Export Instrument Data**.

   The Export Instrument Data dialog box opens.

2. Select the appropriate Lightwright items for export by clicking the **Lightwright** button. (See “Exporting Instrument Data” on page 1703 for more information.) Select **Export field names as first record**. If exporting to version 4 of Lightwright, select **Export in Lightwright 4 format**.

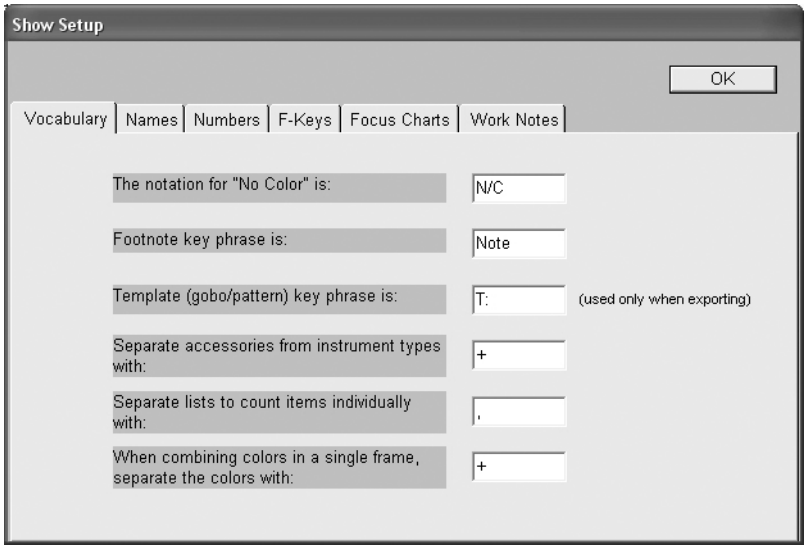
3. Click **Export** to export the lighting instrument information as a tab-delimited file. Specify the location for the exported file, and click **Save**.

Importing the Vectorworks Data into Lightwright

The data exported from the Vectorworks Spotlight program now needs to be imported into Lightwright so that it can be edited.

1. Prior to import, select **Setup > Vocabulary** in Lightwright.

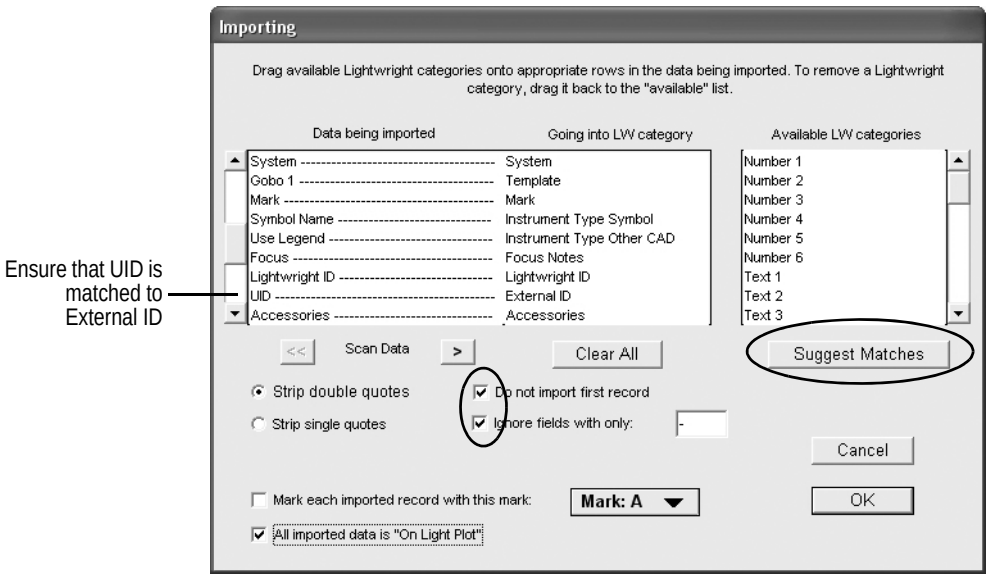
   The Show Setup dialog box opens.



2. Ensure that the **Separate accessories from instrument types with** field contains a "+" sign.

To import Vectorworks data into Lightwright:

1. In Lightwright, select **File > Import Data**. Specify the location of the file that was just created, and click **Open**.
2. The Importing dialog box opens. Click **Suggest Matches**.



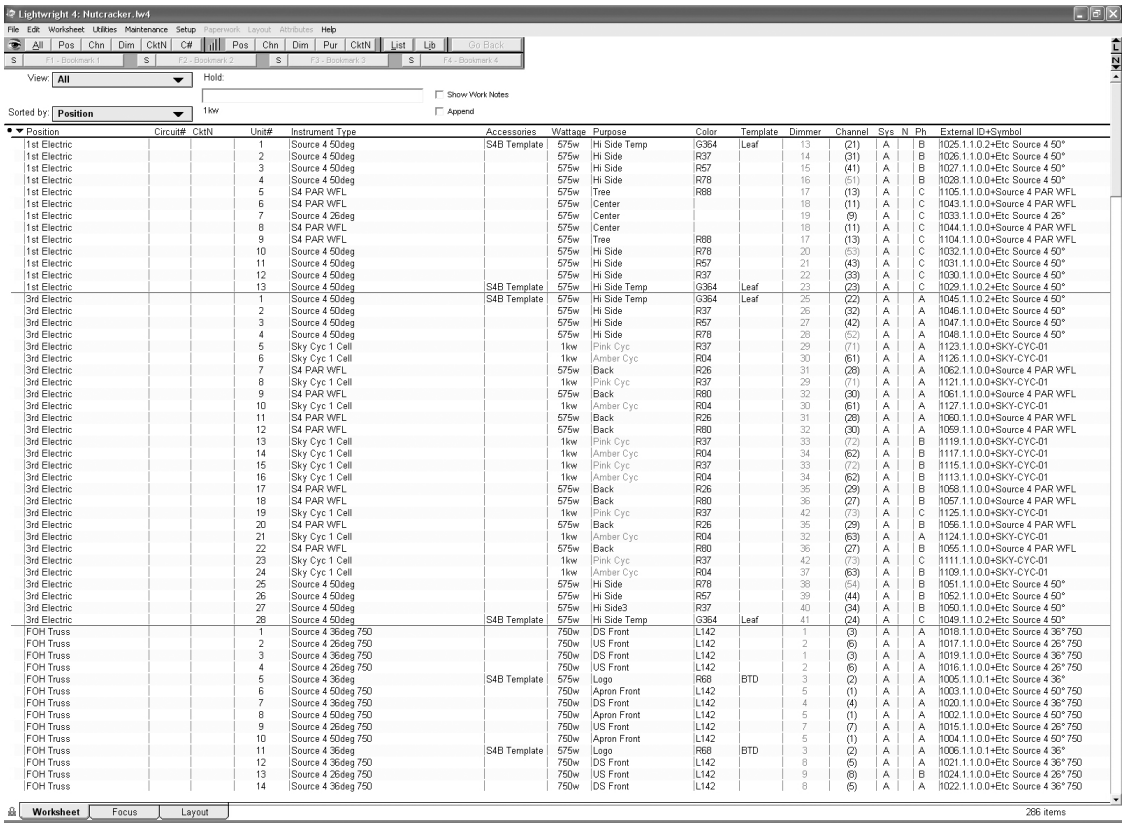
Lightwright attempts to match the imported data fields to Lightwright fields. Verify that the information has been matched correctly.

3. Select **Do not import first record** and **Ignore fields with only**. Enter a "-" (dash) so that Vectorworks Spotlight fields with a "-" are ignored.
4. Click **OK** to begin the import. Lightwright reports the number of instruments that were imported; this may differ from the number of instruments exported from the Vectorworks program due to Static Accessories.
5. Select whether to save the import action in Lightwright.

   Saving the action speeds up the import process, because the matching selections and other options will not need to be specified again.

   Provide a name for the automated action, which can be selected later from **File > Automated Actions**.

6. The instrument information is now available for editing in Lightwright.



Position	Circuit#	CktN	Unit#	Instrument Type	Accessories	Wattage	Purpose	Color	Template	Dimmer	Channel	Sys	N	Ph	External ID+Symbol
1st Electric			1	Source 4 50deg	S4B Template	575w	Hi Side Temp	G364	Leaf	13	(21)	A		B	1025.1.1.0.2+Etc Source 4 50°
1st Electric			2	Source 4 50deg		575w	Hi Side	R37		14	(31)	A		B	1026.1.1.0.0+Etc Source 4 50°
1st Electric			3	Source 4 50deg		575w	Hi Side	R57		15	(41)	A		B	1027.1.1.0.0+Etc Source 4 50°
1st Electric			4	Source 4 50deg		575w	Hi Side	R78		16	(51)	A		B	1028.1.1.0.0+Etc Source 4 50°
1st Electric			5	S4 PAR WFL		575w	Tree	R88		17	(13)	A		C	1105.1.1.0.0+Source 4 PAR WFL
1st Electric			6	S4 PAR WFL		575w	Center			18	(11)	A		C	1043.1.1.0.0+Source 4 PAR WFL
1st Electric			7	Source 4 26deg		575w	Center			19	(9)	A		C	1033.1.1.0.0+Etc Source 4 26°
1st Electric			8	S4 PAR WFL		575w	Center			18	(11)	A		C	1044.1.1.0.0+Source 4 PAR WFL
1st Electric			9	S4 PAR WFL		575w	Tree	R88		17	(13)	A		C	1104.1.1.0.0+Source 4 PAR WFL
1st Electric			10	Source 4 50deg		575w	Hi Side	R78		20	(53)	A		C	1032.1.1.0.0+Etc Source 4 50°
1st Electric			11	Source 4 50deg		575w	Hi Side	R57		21	(43)	A		C	1031.1.1.0.0+Etc Source 4 50°
1st Electric			12	Source 4 50deg		575w	Hi Side	R37		22	(33)	A		C	1030.1.1.0.0+Etc Source 4 50°
1st Electric			13	Source 4 50deg	S4B Template	575w	Hi Side Temp	G364	Leaf	23	(23)	A		C	1029.1.1.0.2+Etc Source 4 50°
3rd Electric			1	Source 4 50deg	S4B Template	575w	Hi Side Temp	G364	Leaf	25	(22)	A		A	1045.1.1.0.2+Etc Source 4 50°
3rd Electric			2	Source 4 50deg		575w	Hi Side	R37		26	(32)	A		A	1046.1.1.0.0+Etc Source 4 50°
3rd Electric			3	Source 4 50deg		575w	Hi Side	R57		27	(42)	A		A	1047.1.1.0.0+Etc Source 4 50°
3rd Electric			4	Source 4 50deg		575w	Hi Side	R78		28	(52)	A		A	1048.1.1.0.0+Etc Source 4 50°
3rd Electric			5	Sky Cyc 1 Cell		1kw	Pink Cyc	R37		29	(71)	A		A	1123.1.1.0.0+SKY-CYC-01
3rd Electric			6	Sky Cyc 1 Cell		1kw	Amber Cyc	R04		30	(61)	A		A	1126.1.1.0.0+SKY-CYC-01
3rd Electric			7	S4 PAR WFL		575w	Back	R26		31	(28)	A		A	1062.1.1.0.0+Source 4 PAR WFL
3rd Electric			8	Sky Cyc 1 Cell		1kw	Pink Cyc	R37		29	(71)	A		A	1121.1.1.0.0+SKY-CYC-01
3rd Electric			9	S4 PAR WFL		575w	Back	R80		32	(30)	A		A	1061.1.1.0.0+Source 4 PAR WFL
3rd Electric			10	Sky Cyc 1 Cell		1kw	Amber Cyc	R04		30	(61)	A		A	1127.1.1.0.0+SKY-CYC-01
3rd Electric			11	S4 PAR WFL		575w	Back	R26		31	(28)	A		A	1060.1.1.0.0+Source 4 PAR WFL
3rd Electric			12	S4 PAR WFL		575w	Back	R80		32	(30)	A		A	1059.1.1.0.0+Source 4 PAR WFL
3rd Electric			13	Sky Cyc 1 Cell		1kw	Pink Cyc	R37		33	(72)	A		B	1119.1.1.0.0+SKY-CYC-01
3rd Electric			14	Sky Cyc 1 Cell		1kw	Amber Cyc	R04		34	(62)	A		B	1117.1.1.0.0+SKY-CYC-01
3rd Electric			15	Sky Cyc 1 Cell		1kw	Pink Cyc	R37		33	(72)	A		B	1116.1.1.0.0+SKY-CYC-01
3rd Electric			16	Sky Cyc 1 Cell		1kw	Amber Cyc	R04		34	(62)	A		B	1113.1.1.0.0+SKY-CYC-01
3rd Electric			17	S4 PAR WFL		575w	Back	R26		35	(29)	A		B	1058.1.1.0.0+Source 4 PAR WFL
3rd Electric			18	S4 PAR WFL		575w	Back	R80		36	(27)	A		B	1057.1.1.0.0+Source 4 PAR WFL
3rd Electric			19	Sky Cyc 1 Cell		1kw	Pink Cyc	R37		42	(73)	A		C	1125.1.1.0.0+SKY-CYC-01
3rd Electric			20	S4 PAR WFL		575w	Back	R26		35	(29)	A		B	1056.1.1.0.0+Source 4 PAR WFL
3rd Electric			21	Sky Cyc 1 Cell		1kw	Amber Cyc	R04		32	(63)	A		A	1124.1.1.0.0+SKY-CYC-01
3rd Electric			22	S4 PAR WFL		575w	Back	R80		36	(27)	A		B	1055.1.1.0.0+Source 4 PAR WFL
3rd Electric			23	Sky Cyc 1 Cell		1kw	Pink Cyc	R37		42	(73)	A		C	1111.1.1.0.0+SKY-CYC-01
3rd Electric			24	Sky Cyc 1 Cell		1kw	Amber Cyc	R04		37	(63)	A		B	1109.1.1.0.0+SKY-CYC-01
3rd Electric			25	Source 4 50deg		575w	Hi Side	R78		38	(64)	A		B	1051.1.1.0.0+Etc Source 4 50°
3rd Electric			26	Source 4 50deg		575w	Hi Side	R57		39	(44)	A		B	1052.1.1.0.0+Etc Source 4 50°
3rd Electric			27	Source 4 50deg		575w	Hi Side3	R37		40	(34)	A		B	1050.1.1.0.0+Etc Source 4 50°
3rd Electric			28	Source 4 50deg	S4B Template	575w	Hi Side Temp	G364	Leaf	41	(24)	A		C	1049.1.1.0.2+Etc Source 4 50°
FOH Truss			1	Source 4 36deg 750		750w	DS Front	L142		1	(3)	A		A	1018.1.1.0.0+Etc Source 4 36° 750
FOH Truss			2	Source 4 26deg 750		750w	US Front	L142		2	(6)	A		A	1017.1.1.0.0+Etc Source 4 26° 750
FOH Truss			3	Source 4 36deg 750		750w	DS Front	L142		1	(3)	A		A	1019.1.1.0.0+Etc Source 4 36° 750
FOH Truss			4	Source 4 26deg 750		750w	US Front	L142		2	(6)	A		A	1016.1.1.0.0+Etc Source 4 26° 750
FOH Truss			5	Source 4 36deg	S4B Template	575w	Logo	R68	BTD	3	(2)	A		A	1005.1.1.0.1+Etc Source 4 36°
FOH Truss			6	Source 4 50deg 750		750w	Apron Front	L142		5	(1)	A		A	1003.1.1.0.0+Etc Source 4 50° 750
FOH Truss			7	Source 4 36deg 750		750w	DS Front	L142		4	(4)	A		A	1020.1.1.0.0+Etc Source 4 36° 750
FOH Truss			8	Source 4 50deg 750		750w	Apron Front	L142		5	(1)	A		A	1002.1.1.0.0+Etc Source 4 50° 750
FOH Truss			9	Source 4 26deg 750		750w	US Front	L142		7	(7)	A		A	1015.1.1.0.0+Etc Source 4 26° 750
FOH Truss			10	Source 4 50deg 750		750w	Apron Front	L142		5	(1)	A		A	1004.1.1.0.0+Etc Source 4 50° 750
FOH Truss			11	Source 4 36deg	S4B Template	575w	Logo	R68	BTD	3	(2)	A		A	1006.1.1.0.1+Etc Source 4 36°
FOH Truss			12	Source 4 36deg 750		750w	DS Front	L142		8	(5)	A		A	1021.1.1.0.0+Etc Source 4 36° 750
FOH Truss			13	Source 4 26deg 750		750w	US Front	L142		9	(8)	A		B	1024.1.1.0.0+Etc Source 4 26° 750
FOH Truss			14	Source 4 36deg 750		750w	DS Front	L142		8	(5)	A		A	1022.1.1.0.0+Etc Source 4 36° 750

7. Save the Lightwright file.

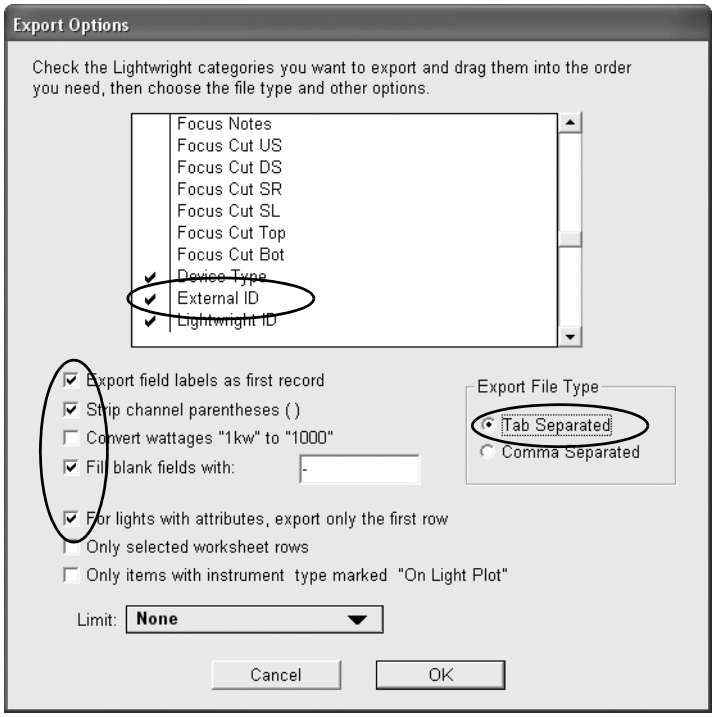
Exporting the Edited Data from Lightwright

Once the lighting instrument data has been edited in Lightwright, export the data so that it can be used to update the light plot in the Vectorworks Spotlight file.

If lighting instruments have been deleted in Lightwright, they will not automatically be deleted in the Vectorworks file. A warning message opens if this is the case, and the affected instruments are automatically selected for further action.

To export the edited data from Lightwright:

1. In Lightwright, select **File > Export > Data**. The Export Options dialog box opens.



2. Select the fields for export; selected fields display a check mark. Ensure that the **External ID** and **Accessory** fields are selected for export, and that Tab Separated is selected as the **Export File Type**. In addition, select **Export field labels as first record**, **Strip Channel Parentheses**, **For lights with attributes** and **Fill blank fields with**. For the blank fields, enter a "-" (dash), so that a "-" is inserted in any blank fields. Click **OK**.
3. Specify the export file name and location.
4. The option to save the export action for future automation is presented.
5. Click **Yes** to save the action.

Importing the Edited Lightwright Data into Vectorworks

Import the edited Lightwright data into the Vectorworks Spotlight file to update the lighting instrument information.

To import Lightwright data back into the Vectorworks file:

1. In the Vectorworks program, select **File > Import > Import Instrument Data**.

   The Instrument Data Import dialog box opens.
2. With the **Browse** button, specify the file exported from Lightwright. If the file to import has External ID or UID as a field name, it will automatically be selected as the UID. Do not select **Import First Record**.

   Click **AutoLink** to correctly match important fields. If not all fields are automatically mapped, map the incoming data to an instrument field as described in "Importing Instrument Data" on page 1704.

   Accessories should be mapped to Accessories, and Instrument Type Symbol to Symbol Name.
3. Click **Import**. The Lightwright data is imported into the Vectorworks file, automatically updating the instrument data in the light plot.

   If the instruments are not updated, verify that the **External ID** field was exported, and the **Export Field Labels as First Record** option was selected in Lightwright.

If the data needs to be sent back to Lightwright after this initial export, select the **Merge** command in Lightwright rather than **Import**. Lightwright uses the Lightwright ID number to match the data to existing instruments rather than creating new instruments.

DXF/DWG and DWF File Formats

DXF and DWG are two common file formats used to exchange information between different CAD and drawing programs. DXF is a semi-public standard promoted and controlled by Autodesk, Inc. DWG is a proprietary, closed format used by Autodesk for its products. DXF and DWG files contain almost identical information, and both store all of the same objects. Recent versions of AutoCAD have issues when reading some DXF files, such as losing links to images, so DWG is usually a safer choice in that case. As of this writing, the most recent DXF/DWG version is version 2015. The Vectorworks program imports versions 2.5 through 2015, and exports versions 12 through 2015.

DWF is a format developed by Autodesk to allow efficient sharing of design drawings with colleagues who do not have access to AutoCAD or other design software. DWF files are highly compressed, device and software independent, and can include 3D models. The Vectorworks program imports and exports versions 4.2, 5.5, and 6.0 (DWF text and binary); 4.2 and 5.5 (DWF compressed binary); 6.01 (3D DWF); and 6.02 (DWFx).

Vectorworks 2015 Improvements

The following changes to the DXF/DWG and DWF translation procedure have been added:

- Standard leader and multileader lines now import as polylines or polygons with arrow markers; custom leader arrows import as symbols. Previously, all leader and multileader lines imported as a set of independent objects.
- New options allow you to import dashed lines as solid lines, and to set a minimum gap size. Previously, even lines with gaps too small to be seen were imported as dashed line types, which could affect the performance of the Vectorworks program.
- The precision settings for length and angle are now set according to the setting in the Units dialog box during export. Previously, only the length precision was set, to a fixed value.
- A new Location tab in the import options allows you to decide how to handle imported objects relative to the drawing's internal origin. The default option centers the drawing about the internal origin, and shifts the user origin, if necessary, to preserve file coordinates. This prevents problems caused by objects located too far away from the internal origin. Subsequent imports match the user origin, keeping all files properly aligned with each other. Other options on the Location tab allow you to preserve the import behavior of previous versions of Vectorworks software.
- A new option allows you to import tables as worksheets, for easier modification. Previously, tables were always imported as sets of individual line and text objects.
- A new option allows you to export SIA dimensions as non-superscript AutoCAD dimensions that update automatically during reshape. Previously, SIA dimensions were always converted into non-dimension objects that looked exactly the same as they did in Vectorworks.

Information Lost in Translation

Importing from and exporting to DXF/DWG or DWF is not the same as saving or opening a Vectorworks file. It is a translation from one way of doing things to another; consequently, information can get lost in the translation. Following are some items that DXF/DWG and DWF handle differently. These can introduce unwanted effects in the translated file.

Item	Description
Units	Version 2000 DXF/DWG and later supports the concept of units, so the file's unit settings can be imported (if it includes units). Previous versions of DXF/DWG do not support units, and some version 2000 and later files may be unitless. There is no way for the Vectorworks program to tell whether these unitless drawings were made in meters, feet and inches, or microns. Communicate with the person providing the file to determine this information. Unitless DXF/DWG files do have five "units" settings (such as "Architectural" and "Engineering") which are used to guess the original units, but the guess may need to be adjusted.
Line Weights	The Vectorworks program allows line weights and colors to be specified independently. AutoCAD has recently gained the ability to do so, but most AutoCAD users still use colors to map to line weights. Version 14 and earlier DXF/DWG files do not support true line weights at all. If you choose to export with line weights mapped to colors, then original object colors will be lost.
Colors and Fills	The Vectorworks program is more graphically rich than DXF/DWG can currently support. In all cases, the Vectorworks program chooses the closest possible translation given the limitations inherent in DXF/DWG. The default version in the export dialog box will always give the best results possible, assuming the recipient's software can read all of the information. DXF/DWG versions prior to 2004 have a fixed color palette (which changes slightly depending on whether the background is black or white) and all objects have just one solid color associated with them. Objects such as circles in DXF have no fill color (just a line color). A few objects can have a fill color, but they have no separate line color. DXF/DWG version 14 and above supports a "solid hatch," which is a separate object that can be associated with objects such as circles to make them look like they have a color fill. These hatches cannot be the same color as the background color (such as a white rectangle on a white background to mask objects underneath). Since these solid hatches can be associative, the Vectorworks program can import them and set the associated object's fill color instead of having two separate objects for frame and fill. DXF/DWG version 2000 supports "wipeout" entities, which are essentially polygonal images filled with the background color. Some AutoCAD users may not want to receive files with wipeouts. DXF/DWG export includes an option to exclude solid fills (which includes both wipeouts and solid hatches). Since wipeouts can only be polygons and are not associative, if a white circle on a white background is exported and then re-imported into a Vectorworks file, the result is an unfilled circle and a polygon with a white fill and no pen inside the circle. The smoothness of the polygon (number of facets) depends on the 2D conversion resolution preference when it was exported. Objects with pattern fills export as a plain solid color. Surface hatches associated with a Renderworks texture must be exported as standard AutoCAD hatches.
Layers and Classes	Each Vectorworks design layer is similar to a DXF/DWG model space. A Vectorworks drawing can have many design layers visible at the same time, with different scales and views for each design layer, but only one model space is allowed in a DXF/DWG file. Therefore, the Vectorworks program has to merge the multiple design layers, and some information can be lost. The drawing should generally look and print the same after the export, but independent layer scales, object coordinates, and invisible objects can be lost.

Item	Description
Groups and Symbols	The Vectorworks program uses symbols, which are objects that can be inserted multiple times without greatly increasing the file size, and which need only one edit to update all copies. It also has groups, which are objects that are grouped together and act as one object. The DXF/DWG equivalent of a Vectorworks symbol is called a “block.” The closest DXF/DWG equivalent to a Vectorworks group is an “anonymous block,” which is like a symbol without a name. Unlike anonymous blocks, named blocks can be edited easily in AutoCAD, and new instances of the blocks can be inserted into the drawing. However, named blocks re-import into Vectorworks files as symbols, which may not be desirable. Therefore, the Vectorworks program has an option to export groups as anonymous blocks, if named blocks cause a problem.
Attributes and Linked Text	DXF/DWG does not have database records that correspond to Vectorworks record formats. It does have objects called attribute definitions (“attdefs”) which, when placed in blocks/symbols, behave somewhat like linked text in a Vectorworks file. The correspondence is very loose and as a result translation of these objects is not always smooth. DXF/DWG block attributes are created for things exporting as blocks (symbols, groups, plug-in objects, or layer links) that have record formats attached. Because only DXF/DWG blocks can have attributes attached, the record format information is not exported for other object types, such as lines or circles.
Names	When exporting to DXF/DWG version 2000 and later, the characters in layer and block names will not change. Accurate translation of lower case text, spaces and other Unicode characters is supported (except for the following illegal characters, which are converted during export: < > " `, / \ : ? * | =). Export to previous versions converts all name characters to uppercase; all spaces and special characters are converted to underscores. DXF/DWG attribute tag names cannot have spaces in any version; any spaces found in record field names are converted to an underscore.
Styled Multiline text	Multiline formatted text includes various sized fonts that are bold, underlined, or italicized, and that wrap to the next line. This type of text is supported for DXF/DWG versions 13 and up. For earlier versions, wrapped text is split into separate lines.
Layer Transfer Mode / Transparency	DXF/DWG does not support layer transfer modes or transparency, so use only paint transfer mode (100 percent opaque on Mac systems and on Windows systems that have GDI+ imaging enabled)
Dimensions	The Vectorworks and AutoCAD programs handle dimensions, units, and dimension standards very differently. Dimensions exported to DXF/DWG look exactly the same when opened in AutoCAD, but they may change appearance slightly if edited. The Vectorworks program also creates appropriate dimension styles for all dimension standards that are used in the file, so even if the recipient modifies the dimensions or creates new ones, they should not look significantly different. During import, the dimension styles from the DXF/DWG file will be created as custom dimension standards in the Vectorworks file, to preserve the original look.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

DXF/DWG and DWF File Import
DXF/DWG and DWF Items Which Cannot Import to Vectorworks
DXF/DWG and DWF File Export
Items Which Cannot Export to DXF/DWG or DWF
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

DXF/DWG and DWF File Import

Consider the following points when importing from DXF/DWG or DWF file format to Vectorworks file format.

Item	Import Notes
Line Weights	If no .ctb file is present and true line weights are present, then the line weights import exactly. If no .ctb file is present and no true line weights are present, values will have to be entered manually. The values in the dialog can be initialized by first importing a file that does have an associated .ctb file. This creates a hidden record that stores the mapping information. A template file could be created after doing such an import, to save the values for future use.
Text	• Text is sometimes unlinked from symbols when imported to preserve the original look. Options are provided for preserving the links instead. • Some text styles are not supported; when the active plane is set to Layer, text in 3D symbols is imported; when the active plane is set to Screen Plane, text in 3D symbols is not imported.
Model Space Entities	Model space entities are imported as design layer objects.
Paper Space and Viewports	Each paper space layout creates a Vectorworks sheet layer when imported. DXF viewports are imported as Vectorworks viewports, with the same scale, view, and projection.
Points	Points are imported as symbols or as loci, depending on the file and the import options chosen.
Multilines	Multilines are imported as grouped lines. An option is provided to import them as walls.
bhatches, Images, and Wipeouts	• bhatch definitions are imported as hatch definitions. • A single object associative bhatch with no islands is imported as an equivalent object with a solid or hatch fill. • A multi-object, non-associative bhatch or associative bhatch is imported as an object with zero line weight and solid or hatch fill. • A bhatch with islands is imported as one or more polylines with holes, with solid or hatch fill. • Images referenced by the DXF/DWG file are imported as Vectorworks images. • A wipeout is imported as a polygon with a solid fill that is the same color as the background.
AEC objects created with AutoCAD Architecture	AEC (architectural, engineering, and construction) objects such as walls, doors, and windows are imported as grouped lines and polygons (2D conversion), or grouped meshes (3D conversion)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Preparing to Import
DXF/DWG and DWF Import Procedure
DXF/DWG and DWF Import Options
DXF/DWG and DWF File Formats
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Preparing to Import

The Vectorworks program imports versions 2.5 through 2015 of DWG and text and binary DXF. Additionally, DWF files of the format .dwf (3D DWF), .dwfx (DWFx), and .w2d (DWF 2D sheet) can be imported. Before import, the following steps are recommended to increase the likelihood of a satisfactory translation:

1. Read "DXF/DWG and DWF File Import" on page 1716 for more information about the differences between the DXF/DWG and DWF formats and Vectorworks file formats.
2. It is not necessary to explode the entire drawing in AutoCAD before importing. If a file is not importing correctly, try exploding individual problem objects before import.
3. If possible, communicate with the file originator. Determine the intended units, page size, and scale of the file, along with the intended color-to-line weight mapping, if any.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

DXF/DWG and DWF Import Procedure
DXF/DWG and DWF Import Options
DXF/DWG and DWF File Formats

DXF/DWG and DWF Import Procedure

There are three commands to import DXF/DWG and DWF files into Vectorworks.

- **Import Single DXF/DWG** and **Import Single DWF** are very similar; they both import a single file into the current file, either directly, or by reference. When a master file is imported with one of these commands, the external reference files ("xrefs") are automatically bound. The Vectorworks program displays an alert if any xrefs are missing, along with the names of the missing files.
- **Import DXF/DWG or DWF** imports one or more files that are in any of these three formats. There are three options for importing the files: import into the current file, which creates a new design layer for each imported file; import objects into the current file as symbol resources; or import each file into a separate Vectorworks file in a designated folder. The results of the import process are summarized in a log file. Referencing is not allowed when multiple files are imported. Also, no alert displays if xrefs are missing when multiple files are imported.

Verify that the best possible options were selected for import by checking the imported file(s). If the import was not successful, try again with different options. Import options can be saved as a set.

> Sometimes files import with large numbers of small lines that are connected end to end. To create a single object, first use the **Edit > Select Connected Objects** command to select all connected lines. Then use the **Modify > Compose** command to compose them into one object. See "Selecting Connected Objects" on page 1021, and "Composing and Decomposing Objects and Surfaces" on page 1020.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing DXF/DWG and DWF Files
Importing a Single DXF/DWG or DWF File

Importing DXF/DWG and DWF Files

When you have several DXF/DWG or DWF files to import, use the **Import DXF/DWG or DWF** command for maximum flexibility.

To import DXF/DWG or DWF files:

1. Create a blank file and set the drawing size, or open an empty template that already has the correct drawing size.

   Import into an existing Vectorworks drawing is not recommended, because it can produce unexpected results. For the same reason, if you import more than one file with the **Import DXF/DWG or DWF** command, ensure that the files are very similar.

2. Select **File > Import > Import DXF/DWG or DWF**.

The Import dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Source	
One or More Files	Imports only one file or only certain files within a folder. Click **Choose Files** and choose one or more DXF/DWG or DWF files to import. The number of files selected and their location is displayed.
All ___ Files in Folder	Imports all files of the selected type (DXF only, DWG only, DXF and DWG, or DWF only) from a specified folder. Click **Choose Folder** and choose the source folder.
Include Subfolders	If **All Files in Folder** is selected, includes all files of the selected type in all subfolders
Destination	
Current File	Imports the selected file(s) into the current file. If several files are selected, a new layer is created for each imported model space.
Symbols in Current File	Imports the selected file(s) into the current file. Each file creates a separate symbol. This is convenient when importing part catalogs, for example. Paper space is not imported.
New Files in Folder	Converts the selected file(s) into new, separate Vectorworks files in the selected folder. Click **Choose Folder** to choose the location. This is the best option when converting many files.
Use Document Template	If **New Files in Folder** is selected, select a template to use for each new file, or select Blank Document. The template can be useful for specifying the page size to use for model space objects (which can also affect dash scales and conversion of polyline widths). It also allows specification of default color to line weight mappings, and in some cases, units.
References	
Bind External References	Imports the selected master DXF/DWG file and all its external references into the current file
Ignore External References	Imports the selected master DXF/DWG file without its external references
Use Design Layer Viewports (Vectorworks Design Series required)	If **New Files in Folder** is selected, imports the selected master DXF/DWG file and all its external references into separate Vectorworks documents; the external documents are referenced from the master Vectorworks document using design layer viewports
Use Layer Import	If **New Files in Folder** is selected, imports the selected master DXF/DWG file and all its external references into separate Vectorworks documents; the external documents are referenced from the master Vectorworks document using layer importing
Architectural Objects	
2D View	Imports AEC objects created with AutoCAD Architecture (such as walls, doors, and windows) in 2D view; only 2D graphics are imported
3D View	Imports AEC objects in 3D view; only 3D graphics are imported

Parameter	Description
Import Options	
Use Settings	Select a set of import options to apply to the imported files. If several files will be imported, it is recommended that **Units Setting In File** be set to Determine Automatically. • The Default options are those that ship with the Vectorworks software, and they cannot be changed. • To select custom options and save them as a set, select <Active Settings> or a set name, and then click **Set Custom Options** to open the DXF DWG Import Options dialog box. • To use custom import options that have been saved, select the set name from the list.

3. Click **OK** to import the file(s).

   The progress of the import is displayed for each imported file during import. The final results of the import process are shown in the DXF/DWG or DWF Import Results dialog box.

4. Click **Details** to open the results log file. The log text file, named DXF_DWG_DWF Import Log, is placed in the specified destination folder if it exists, or in the Vectorworks user folder. New log information is appended to any existing log file. The report provides a summary of the import, including the import settings that were used, and a list of which files succeeded, and which failed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing a Single DXF/DWG or DWF File
DXF/DWG and DWF Import Options

Importing a Single DXF/DWG or DWF File

To import a single DXF, DWG, or DWF file:

1. Create a blank file and set the drawing size, or open an empty template that already has the correct drawing size.

   Import into an existing Vectorworks drawing is not recommended, because it can produce unexpected results. For example, the class attributes for existing objects in the drawing may be overwritten. Workgroup referencing may give better results in these cases.

2. Select **File > Import > Import Single DXF/DWG** (or **Import Single DWF**).

   The Import DXF/DWG Files (or Import DWF Files) dialog box opens. Select the file to import; the progress of the initial import is displayed.

   Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

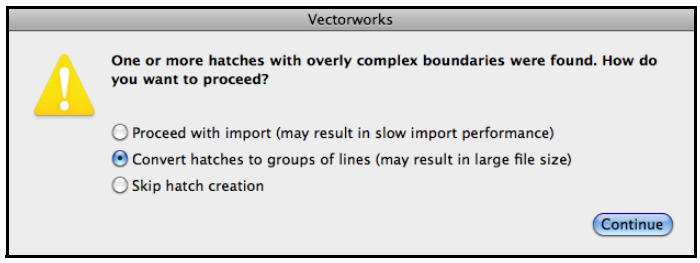
3. During the import, the import options dialog box opens.

   Select appropriate import options as described in “DXF/DWG and DWF Import Options” on page 1720.

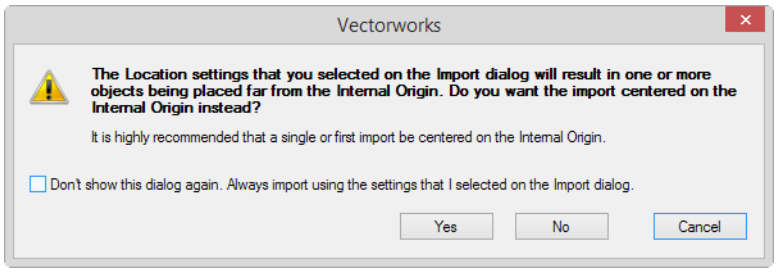
4. Click **OK** to continue importing the file. A progress bar displays, along with the number of objects processed and free memory available.

5. If the file being imported has one or more complex hatches with multiple boundaries, the import may have performance problems. An alert displays to allow you to choose how to process the hatches. Select an option and click **Continue**. If you select the option to convert hatches to groups of lines, solid type hatches will be skipped rather than imported.

   For batch imports, no alert displays; the last option you selected from this dialog box for single file import is used by default.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



6. The Location tab of the import options has a default setting, which provides the best chance of locating the imported data close to the internal origin, while keeping subsequent imports aligned: **Center first import, align all subsequent imports**. If the file being imported contains objects located far from the internal origin, and this option was deselected in the import options, an alert opens to warn you of potential issues.



   Select **Yes** to switch to the option to **Center first import, align all subsequent imports**. Select **No** to continue the import with the option that was selected on the Location tab.

7. A results dialog box displays to notify you whether the import succeeded or failed.
8. Click **Details** to open the results log file. The log text file, named DXF_DWG_DWF Import Log, is placed in the specified destination folder if it exists, or in the Vectorworks application folder. New log information is appended to any existing log file. The report provides a summary of the import, including the import settings that were used, and a list of which files succeeded, and which failed.
9. Check the imported file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing DXF/DWG and DWF Files

DXF/DWG and DWF Import Options

The DXF DWG Import Options dialog box and the DWF Import Options dialog box are very similar. This is because when you import from DWF, the drawing objects are first converted to DXF/DWG, and then converted into Vectorworks format. Change the import options as needed to import a DXF/DWG or DWF file with the maximum integrity. The same import options display for import of single files and batches of files.

The first tab, Primary Settings, contains important basic settings. The Graphic Attributes and Objects tabs contain settings for advanced users to customize the import process. The Location tab helps with importing a file and locating it appropriately relative to the internal origin of the Vectorworks drawing.

Options that do not apply to the current file import appear dimmed, and an explanation displays next to the option. For example, if no points are contained in the file to be imported, the Points options are dimmed, and the message “No points were found” displays.

If true line weights are not present in the file, you can map the DXF/DWG line colors to Vectorworks line weights for import (on the Map Colors to Line Weights dialog box).

To make custom imports faster and easier, save the sets of options that you use frequently.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Import Options: Primary Settings Tab

The options on the Primary Settings tab establish the basic import options.

Click to show/hide the parameters.

Parameter	Description
Saved Settings Options	
Settings	Select <Active Settings> to use the import options currently selected in the dialog box; to use a set of saved import options, select them from the list
Save	Opens a dialog box to name and save the currently selected import options so that they can quickly be selected as a set
Manage	Opens the Saved Settings dialog box to rename or delete sets of saved import options see “Managing Saved Import Options” on page 1729
Model Space Units	Version 2000 and later DXF/DWG or DWF files support true units. When importing a file with true units, the Vectorworks program determines and sets the units automatically. If the imported file does not contain true units information, the program tries to guess the units setting; however, it may still require adjusting (see “Setting Units Manually” on page 1723).
Units Setting In File	The Vectorworks program tries to determine the imported file’s unit settings, and displays the information to the right of **Units Setting in File**, along with the unit format (such as Architectural), and the scaling factor that will be assumed (such as 1 **DXF/DWG Units** = 1”). • To use the displayed unit settings in the Vectorworks file, select Determine Automatically from the list, and also select **Set Vectorworks Units to Match**. • To use another unit type (such as Centimeters), select it from the list, and also select **Set Vectorworks Units to Match**. • To use a custom unit type, or to change the defaulted scaling factor, select Custom from the list, and then specify the units in the text boxes. Enter the unitless DXF/DWG or DWF number in the first box, and a number with units in the second box. (For example, 15 DXF/DWG Units = 1” in the Vectorworks drawing.) The units displayed in the second box are the same as the current Vectorworks document settings, but other units can be entered as long as the appropriate units suffix is included, such as 3 cm. Also select **Set Vectorworks Units to Match**. • To use the units currently set in the Vectorworks file, select Use Vectorworks Document Units. The Vectorworks program assumes that 1 DXF/DWG or DWF unit = 1 current document unit when importing objects.
Set Vectorworks Units to Match	Changes the Vectorworks document units and units format to match the DXF/DWG or DWF file being imported; the physical sizes of imported objects will not be affected. This option is disabled when Use Vectorworks Document Units is selected from the **Units Setting In File** list.
2D/3D Conversion	

Parameter	Description
Convert Objects To	Specify whether objects should be imported as 3D, 2D, or a mixture of 2D and 3D. Generally, select the 2D and 3D option, which converts objects that appear to be 2D (planar objects parallel to or in the active layer plane) to Vectorworks 2D objects. The remaining objects are imported as 3D. To import 3D versions of AutoCAD Architecture objects such as walls and doors, select the **3D View** setting for Architectural Objects on the DXF/DWG or DWF Import dialog box, and select a 3D conversion option. To import both 2D and 3D versions of the objects, import them twice from the DXF/DWG file: once with **3D View** selected, and once with **2D View** selected. Because the Vectorworks program does not have 3D text, selecting 2D and 3D can cause text in 3D symbols to be deleted. When the active plane is set to Layer, text in 3D symbols is imported; when the active plane is set to Screen Plane, text in 3D symbols is not imported. In addition, objects parallel to the active layer plane that have a thickness are imported as 3D even though the originator of the file may not have intended for them to be 3D. If problems occur, import all objects as 2D by selecting All 2D. A warning displays if selecting this option will distort any objects, such as 3D symbols with 3D rotation. If the file contains only a 3D model, select the All 3D option. Otherwise, parts of a large object composed of several entities could be converted to 2D. Occasionally, none of the choices is appropriate for all the objects. In this case, select the option that best converts most of the objects.
2D/3D Conversion Res	Specify the resolution that will be used to import both 2D and 3D objects. By default, the 2D and 3D resolutions set in the Vectorworks Preferences are used (see "Edit Preferences" on page 49 and "3D Preferences" on page 54). Higher resolutions can have an adverse effect on drawing performance. This parameter is disabled for DWF import.
Model Space	Once the units have been determined, specify the scale to display the imported file. Choosing the model space scale is important. The scale affects the dash length scaling and the conversion of polylines with widths (world-space line weights) to Vectorworks line weights. If the scale or drawing size are set incorrectly, some polylines may seem to have the wrong line weight and some dashes may be too long or too short.
Fit to Page	The Vectorworks program estimates a scale based on the bounds of all of the objects in model space; the scale fits those objects on the page. Select this option to use the estimated value.
This Scale	To import at a different scale, set the scale manually. Click **Scale** to open the standard Layer Scale dialog box, and select the desired scale. (**All Layers** and **Scale Text** do not apply to DXF/DWG files.) Click **OK** to return to the Primary Settings tab, and the selected scale displays.
Reference	References the original DXF/DWG or DWF file (master file), so that the imported objects can be updated when the master file changes. See "DXF/DWG and DWF References" on page 1728 for more information about referencing. This option is unavailable if multiple files are being imported.
Absolute path	Stores the absolute file path of the master file. Select this option when the location of the master file will not change in the future, or if the master file is on another volume.

Parameter	Description
Path relative to current document	Stores the file path of the master file relative to the target Vectorworks file; this option is available only if the master file is on the same volume as the target file. Select this option if the target file and the master file may be moved to another volume in the future.
Save referenced cache to disk	Saves a copy of the referenced data with the target Vectorworks file. When this option is deselected, a copy of the referenced data is not saved, which means that the target file size is smaller; the referenced data is updated when the target file is opened
Automatically update out of date reference during file open	Updates the reference each time the target Vectorworks file is opened; when deselected, the reference is updated only when **Update** is clicked from the References tab of the Organization dialog box.

Sometimes DXF/DWG or DWF drawings are split up into pieces and saved as separate files, such as different areas of a large city map. When multiple files like this are imported into one file, they need to have their coordinates aligned, and their layer scale set the same. The recommended workflow is to import the files with the **Center first import, align all subsequent imports** option on the Location tab.

The Primary Settings tab covers the basic requirements for importing a DXF/DWG or DWF file. If the results are not satisfactory, explore the options on the Graphic Attributes and Objects tabs.

Setting Units Manually

If objects seem to be the wrong physical size after import, ensure that the units chosen are correct. (**Model Space Scale** only affects the display, but **Units Setting in File** affects the actual measured size of the objects.) DXF/DWG or DWF files do not always have the true units set, and sometimes have incorrect units set.

The Vectorworks program guesses the units based on the information available, and indicates what it found in the dynamic text at the top of the pane. If the guess is wrong, set the units manually.

If you do not know the correct units, but you know the true length of one of the objects in the drawing, determine the true units as follows.

Import the file and choose Custom units, setting the edit boxes to something like 1 **DXF Units** = 1". After import, measure the size of the object that you know the true length of. Close the document and redo the import, but this time set the units to Custom with these values in the edit boxes: (measured length) **DXF Units** = (true length). For example, if the true length is 1", but the measured length is 2.54", enter 2.54 **DXF Units** = 1". (Do not include units in the first box, and if in feet and inches mode, just use the total measured length in inches.)

If the Vectorworks program finds an exact match for that ratio, it will change the Custom choice to the correct units. (In the example above, it changes it to Centimeters.) If the measurements and the ratio are not exact (for example, 2.539 instead of 2.54), manually adjust it to a standard ratio. Common ratios have values such as 1, 12, 2.54, and powers of 10. Examples: 1/12, 12/10, 2.54/0.01, etc.

If you do not know any true lengths, but the document contains dimension objects that show lengths, follow the steps just described with the following change: Select **Convert Dimensions to Groups** (see “Import Options: Objects Tab” on page 1725), import, and use the value in the imported dimension object as the true length.

Normally the Vectorworks program leaves the document units setting unchanged after an import. If a DXF/DWG or DWF file is set to meters, but the Vectorworks document is set to feet and inches, then a dimension object that shows 1 meter in the original file will show as 3'3.37" in the imported file. If you want the imported drawing to be the same as the original, select the **Set Vectorworks Units to Match** option. You can also import with the option to convert dimensions to groups (see “Import Options: Objects Tab” on page 1725) if you want the document to stay in the current units, but you want the dimensions to look the same as in the original.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

DXF/DWG and DWF References
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Import Options: Graphic Attributes Tab
Import Options: Objects Tab
Import Options: Location Tab

Import Options: Graphic Attributes Tab

The Graphic Attributes tab contains settings for advanced users to customize the import process.

Click to show/hide the parameters.

Parameter	Description
Color and Line Weights	Most DXF/DWG or DWF files do not use true line weights, although that is starting to change. Color is traditionally used to indicate line weights. There are some standards that specify mappings between colors and line weights, but those standards are often ignored. The Vectorworks program automatically chooses the standard mappings when exporting, if the document does not already have a hidden record left over from a previous export or import that specifies the mappings. When importing, the Vectorworks program does not choose the standard mappings, but rather defaults to a reasonable line weight for all colors, implicitly alerting the user that they should determine the true mappings that are desired. If desired, specify mappings by selecting **Map Colors to Line Weights**. Select the desired color mapping during the import process; if necessary, communicate with the file originator to determine the correct line weights. The colored lines can be changed to black (or white if the background is black). Select **Set Colors to Black and White**. Note that most DXF/DWG or DWF files are created with a black background, and the colors may not show up well on a white background. Version 2000 and later uses .ctb files to store color mapping information, but it also supports line weights, so these options may or may not be needed. • If no .ctb file is detected during import, and true line weights are present, **Map Colors to Line Weights** is deselected automatically, and the line weights import exactly. • If no .ctb file is detected, and true line weights are not present, **Map Colors to Line Weights** is selected automatically. The Map Colors to Line Weights dialog box displays to allow manual mapping (by default, all colors are mapped to the same line weight, or to the last mapping used during the current session). If a .ctb file is detected, **Map Colors to Line Weights** is selected automatically, and the Vectorworks program reads the file to determine how colors should map to line weights. A dialog box displays to allow manual mapping (values are pre-set by the mapping file; duplicate mappings are indicated by italics). If you used a set of saved import options, the .ctb file overrides any color mappings in your saved settings. Remove the .ctb file from the DXF/DWG or DWF file folder to use the saved settings instead.
Dash Patterns	In some files, dash lengths may import at an inappropriate scale. Change the dash length scale by selecting **Manually Scale All Dash Lengths by** and entering a scale value. A suggested scale value is displayed. The scaling does not affect line thickness. The Vectorworks program converts any dash patterns that are extremely small to solid lines to avoid unacceptable slowdowns during file display and printing. To adjust which dash patterns convert to solid lines, select **Import as Solid Line when Gap is Smaller than** and enter the smallest line gap size you wish to maintain (in document units).

Parameter	Description
Classes/Layers	DXF/DWG and DWF layers correspond more closely to Vectorworks classes than they do to Vectorworks layers. Normally, import DXF/DWG and DWF layers as Vectorworks classes. There is no equivalent to Vectorworks layers in a DXF/DWG or DWF file. When DXF/DWG or DWF layers are mapped to Vectorworks layers, a warning displays if some of the objects in symbols or groups would leave their original layers and take on the layer of the symbol or group. For simple files without blocks, or files with objects inside the block that are set to "by block" attributes, importing layers as Vectorworks layers should pose no problems. To group imported DXF layers in Vectorworks class and layer lists for easy identification, select the option to add a prefix to the imported layers. Enter the custom prefix to use in the field to the right of the check box. Depending on whether you are importing layers as classes or as layers the prefix and hyphen are added as appropriate to either the beginning of the layer or class name. Rays and Xlines (or construction lines) are DXF/DWG or DWF objects that are similar to Vectorworks guides. A ray starts at a point and goes off to infinity, while a construction line is anchored at a point and goes off to infinity in both directions. Select **Rays and XLines Use Guides Class** to convert Rays and Construction Lines into lines in the Guides class, which are of finite length.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Import Options: Primary Settings Tab
Import Options: Objects Tab
Import Options: Location Tab
DXF/DWG and DWF Import Options
Managing Saved Import Options

Import Options: Objects Tab

The Objects tab contains settings for advanced users to customize the import process. In particular, decisions about preserving object visibility and record field links are available.

Click to show/hide the parameters.

Parameter	Description
Points	DXF/DWG or DWF Points act either like a locus in a Vectorworks file (a drawing aid that is not printed), or like a symbol that is visible and can be printed. Select whether to import points as **Loci** or as **Symbols**. The default behavior is to import as symbols if the point style has specified a real world size (that is, it is zoom invariant and always prints the same), or as loci if the point style is specified as a percentage of the pixel size of the drawing window (that is, it varies with the zoom level). Select **Use Guides Class** to import points as **Symbols** in the Vectorworks Guides Class; this allows them to be hidden. In the Vectorworks program, use the **Show/Hide Guides** commands to toggle their display.
MultiLines	Multilines (or "mlines") are similar in some ways to walls in a Vectorworks file, in that they can have breaks that are "healed" and can be connected to other multilines. There are several significant differences between multilines and walls, but an option is provided to import multilines as walls. Do not select this option unless all multilines in the DXF/DWG or DWF file are intended to be walls. When you import multilines as walls, the wall heights may need to be adjusted. To do so, select **Tools > Custom Selection**, select all walls, and change their height in the Object Info palette.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Tables	Select **Grouped Text and Lines** to import each table as a group of independent objects and text that will precisely reflect the format and structure of the original table; select this option if you only need the graphical representation of the table and do not intend to alter the table or its content. Select **Worksheets** to create a worksheet resource and place a worksheet object in the drawing. The worksheet has the attributes of the original table and can be more easily modified.
Block Attributes	"Attributes" in a DXF/DWG or DWF file are similar to linked text in a Vectorworks file. Select how to handle attribute conversion into Vectorworks symbol linked text. Regardless of the selection, record formats will be created based on the attributes found during import, and imported blocks with attributes will have record formats attached. Choose a **Linked Text Handling** method from the list. Depending on the file to be imported, not all options are available.
Preserve Look and Links	This option should be selected when available. The attributes will display as they did in the DXF/DWG or DWF file, and they will be converted to symbol linked text. If the file contains invisible attributes or attributes that override the attribute definitions, this option is not available.
Preserve Look	This is the next best option. The attributes will look and print as they did in the DXF/DWG or DWF file, but some symbol text may become unlinked, if necessary, to preserve the look. Symbol record field updates are not reflected in unlinked text. With this option, an individual symbol could have a mixture of linked and unlinked text.
Preserve Visible Links	This option is available when the DXF/DWG or DWF file contains attributes marked as "invisible." This type of attribute is converted in to a record format attached to a symbol, but is not converted to linked text. The look of the drawing should be preserved, but minor changes to visibility are possible. The advantage of this method is that linked text is preserved for all visible attributes.
Preserve All Links	The link between record fields and symbol text is preserved with this option. Any invisible attributes will become visible. The look of the DXF/DWG or DWF file may not be preserved, but the symbol text will not be unlinked.
Hide All	All attribute text is hidden upon import. The information is still attached to the object in record format, but it is not visible.
Group Record Fields	Each block attribute imports as a record format field. Select **Group Record Fields** to group those fields into a single record format. The Vectorworks program groups sets of attributes into record formats if they have the same set of field names. For example, if two blocks (symbols) have attributes with field names of "color" and "part number," both blocks are converted with the same record format. If one of the blocks has "color," "part number," and "price," while the other only has two of those, then the blocks are converted with different record formats. When grouped, the record format name is created from the names of the first few fields in the record format. Deselect **Group Record Fields** to create one record format per field, with no grouping. Without grouping, the record format uses the same name as the record field.
Blocks	

Parameter	Description
Ignore Block Clipping	Select **Ignore Block Clipping** to import a clipped block as a cropped design layer viewport (if Vectorworks Design Series is installed) or as a cropped layer link. This can create many layers in the Vectorworks file, and the created objects cannot be manipulated easily. Deselect **Ignore Block Clipping** to import a clipped block as a normal scaled or unscaled Vectorworks symbol, which may look very different from the original image.
Dimensions	
Convert Dimensions to Groups	By default, the import process automatically creates custom dimension standards to match the dimension styles in the DXF/DWG or DWF file. If a dimension was customized in the DXF/DWG or DWF file after the style was applied, the converted dimensions may not match the originals; select **Convert Dimensions to Groups** to convert the dimensions into groups instead.

Import Options: Primary Settings Tab
Import Options: Graphic Attributes Tab
Import Options: Location Tab
DXF/DWG and DWF Import Options
Managing Saved Import Options

Import Options: Location Tab

Depending on the coordinates of the imported file, objects in the file might be located far from the drawing's internal origin, which can cause issues with OpenGL rendering and with calculation precision due to rounding errors. It is best to center the first import about the internal origin at import, and align subsequent imports with the user origin. This is the recommended option. Normally, the internal origin and the center of the page are coincident, so this option also centers the objects to the page center.

Click to show/hide the parameters.

Parameter	Description
Center first import, align all subsequent imports	Centers the first imported file about the Vectorworks drawing's internal origin, preventing problems with the imported drawing's coordinates. The file's user origin is shifted if necessary, so that all coordinates and measurements appear unchanged. Any subsequently imported files are then aligned to the file's user origin, so that the imported files are aligned correctly relative to each other.
Center on Internal Origin	Centers the imported file about the Vectorworks drawing's internal origin. The file's user origin is shifted if necessary, so that all coordinates and measurements appear unchanged. Only objects from this import may have their user origin shifted by a specific distance, and potentially, previously imported objects and subsequently imported objects may have their user origin shifted by a different distance. Objects from these other imports may not be at the same relative user origin location.
Align with User Origin	Imports objects relative to the user origin currently set in the Vectorworks file. Note that some imported objects may display off of the page. If objects are located far from the Vectorworks drawing's internal origin, calculation issues may result.
Align with Internal Origin	Positions imported objects according to the Vectorworks drawing's internal origin, but does not shift the user origin; coordinates of the imported file may change.

Internal Origin and User Origin
Import Options: Primary Settings Tab
Import Options: Graphic Attributes Tab
Import Options: Objects Tab
DXF/DWG and DWF Import Options
Managing Saved Import Options

DXF/DWG and DWF References

When you import a single DXF, DWG, or DWF file, you can create a reference to the original file, if the original file may change, and you want to keep the imported objects up to date.

You can create the reference during the import process, or create it from the Organization dialog box. The Organization dialog box is also used to edit, update, and delete references. See “Workgroups and Referencing” on page 207 for more information.

This type of reference is similar to the type of reference that you can create to a design layer in another Vectorworks file. You can import from the referenced file into a design layer, or into a design layer viewport (if Vectorworks Design Series is installed). The import method is set in the Organization dialog box.

After the reference is created, you may need to edit it. To edit references from the Organization dialog box, select the reference and click **Edit**. The Edit Reference dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Source File	Displays the path and file name of the referenced master file; click **Browse** to edit the file location
Save reference location as	Maintains either an absolute or relative file path reference from the current file to the referenced file. Use the absolute path when the location of the referenced file with respect to the current file is not going to change. Use the relative path when the files might be moved to another computer or platform; as long as the relative path between the files remains the same, the reference can be found. Both files must be saved on the same volume to select this option. The **Source File** path displays either an absolute or relative path, depending on the selection.
Save referenced cache to disk	Saves a copy of the referenced data with the target file. When this option is deselected, a copy of the referenced data is not saved, which means that the target file size is smaller; the referenced data is updated when the target file is opened.
Automatically update out of date reference during file open	Updates the reference each time the target file is opened; when deselected, the reference is updated only when **Update** is clicked from the References tab of the Organization dialog box
Import Settings	Opens the Import Options dialog box, to adjust the settings as needed. The next time the reference is updated, the new import settings will be used.

If the reference is a design layer viewport, you can also edit the viewport as follows (also see “Modifying Viewports” on page 1648):

- From the Object Info palette, edit the classes and layers shown in the viewport.
- Right-click (Windows) or Ctrl+click (Mac) the viewport and select the **Edit Crop** command to edit the viewport crop.

DXF/DWG and DWF Import Options

Managing Saved Import Options

If you have saved a set of import options, use the **Manage** button on the import options dialog box to rename or delete these saved settings when needed.

To manage saved DXF/DWG or DWF import settings:

1. Select **File > Import > Import Single DXF/DWG** (or **Import Single DWF**), and select a file to import. The appropriate import options dialog box opens.
2. Click **Manage**.

   The Saved Settings dialog box opens. Select the set of saved import options to change.

Click to show/hide the parameters.

Parameter	Description
Rename	Select and enter a new name for the set of options. If the entered name is already assigned to another set of options, you are prompted to confirm that you want to replace the existing set with the set being saved.
Delete	Select to delete the set of options. You are prompted to confirm that you want to delete the saved settings.

3. Click **OK** to close the Saved Settings dialog box. Click **OK** again to close the import options dialog box.

DXF/DWG and DWF Import Options

DXF/DWG and DWF Items Which Cannot Import to Vectorworks

Certain DXF/DWG and DWF items have no equivalent in a Vectorworks file and cannot be imported. A few other objects theoretically could be imported, but the Vectorworks program does not currently support them.

- Named views
- Textures
- Lights
- rtext
- SHX fonts
- User coordinate systems (UCS)
- Custom line styles and shape files
- DXF groups (named selection sets, not the same as Vectorworks groups)
- Certain objects from industry-specific variations of AutoCAD, if those objects do not have proxy graphics saved in the file. AEC objects that were created with AutoCAD Architecture can be imported as groups of 2D or 3D objects, depending on how you import; for example, a door imported from a DWG file with a 3D view is converted to a group of 3D mesh objects in a Vectorworks file.

DXF/DWG and DWF Import Options

DXF/DWG and DWF File Export

Consider the following points when exporting from a Vectorworks file to DXF/DWG or DWF.

Item	Export Notes
Linked Text	Linked text in symbols export as block attributes.
Symbols, Plug-ins, Layer Links, and Groups	Symbols, plug-ins, layer links, and groups export as blocks. Blocks for symbols, plug-ins, and layer links are given a generic name such as “Group-2” unless they were named in the Data tab of the Object Info palette. By default, groups also are exported as named blocks, but there is an option to export them as anonymous blocks.
Line Weights	By default, line weights are converted to the closest DXF line weight. If the color mapping option is used, mapping information is entered during export and written to a .ctb file.
Fills, Patterns, Gradients, and Hatches	Solid fills and patterns export as DXF objects with associated solid hatch or unassociative wipeout. Wipeouts are only available in version 2000 and later. Hatches export as associative hatches to AutoCAD version 14 and higher, or as anonymous blocks to AutoCAD version 13 and lower. There is an option to export all of the fills, patterns, gradients, and hatches within a class or layer to a separate DXF layer.
Raster Images	Raster images export to the same folder that the drawing and any hatch pattern files are exported to. They export as JPEG files, with an image object in the DXF/DWG file that stores the name of the JPEG file and the insertion point. Images are only supported in AutoCAD versions 14 and higher.
Solids and NURBS Surfaces	Solids and NURBS surfaces typically export as ACIS objects. The following objects cannot be exported as ACIS objects: meshes (which export as DXF meshes), 3D polygons (which export as 3D polygons, or as triangles if filled), NURBS curves (which export as DXF splines), and walls (which export as triangulated 3D polys in 3D views, and as lines and arcs in 2D views).
Page-based symbols, world-based symbols	Page-based symbols are typically used for annotation, and export as annotative blocks with the annotative property set to “true.” World-based symbols export as blocks with the annotative property set to “false.”

Item	Export Notes
Design Layers, Sheet Layers, and Viewports	• Vectorworks design layers export as DXF model space entities. If only design layers (no sheet layers) are exported, individual design layers' scales and views are exported to model space as one scale and view. There is an option to export all of the design layers in a file as separate DXF/DWG or DWF files. • By default, each design layer viewport (Vectorworks Design Series required) is exported to a referenced DWG file; cropped viewports become clipped referenced files, and nested viewports become nested referenced files. This preserves the visibility overrides from the original viewport, because the visibility of the layers in the referenced file can be controlled separately from the layers in the master file. There is an option to export design layer viewports as part of the master document instead. • By default, Vectorworks sheet layers export as DXF paper space layouts, with normal viewports exported directly, and section viewports exported as blocks. Annotations become paper space objects on top of the viewports. AutoCAD requires all paper space objects to be either above or below the viewports, but not both, so any objects or annotations that appear below the viewports in the Vectorworks file will appear above the viewports in the AutoCAD file. Therefore, avoid putting anything underneath viewports in your Vectorworks files. • If sheet layers are exported, all design layers referenced from viewports on the sheet layers are exported to model space in top view without scaling, and viewports are created in paper space layouts to show the various views and scales. Sheets should generally look and print as expected (though without rendering modes set due to some AutoCAD bugs), but the single model space may have many overlapping objects and may not be as usable. There is an option to export the selected sheet layers as separate DXF/DWG files. In addition, there is an option to export sheet layer viewports as 2D graphics to model space (each layer as a separate file). • Since AutoCAD has only one model space, and it does not have hybrid 2D/3D objects, it is sometimes necessary for the Vectorworks program to export the design layers as one or more blocks with special DXF layer names to control the block visibility. For example, a design layer called "Foundation" might export as blocks named "Foundation (2D)" and "Foundation (3D)," with associated DXF layers named "_Foundation (2D)" and "_Foundation (3D)." The blocks will be inserted in model space and assigned to those DXF layers. Viewports that should hide either the 2D or 3D objects will freeze or thaw the appropriate DXF layer. • When you export sheet layers, the Vectorworks program optimizes the translation to preserve the look of the sheets, at the cost of possibly adding some complexity and decreasing the ease of editing items in model space. To minimize the complexity, export sheets with unrelated design layers as separate files, or limit the drawing to one Vectorworks design layer.

Preparing to Export
DXF/DWG and DWF Export Procedure
DXF/DWG and DWF Export Options
DXF/DWG and DWF File Formats

Preparing to Export

The following procedures are recommended to help ensure a satisfactory translation.

1. Export behaves differently based on whether design layers or sheet layers are selected for export.
   - If one or more sheets are selected for export, a paper space layout is exported for each sheet with the sheet objects; any design layers visible in sheet layer viewports are exported to model space. (Sheets that do not share design layers should usually be exported to separate files.) Other sheet layers and unreferenced design layers are omitted from export.
   - If only design layers are selected for export, the design layers are exported to model space and all sheet layers are omitted. If the design layers are exported to a single file, only the visible layers are exported; if the layers are exported to separate files, all layers are exported.
2. AutoCAD does not have multiple model spaces. If the sheet layer being exported has multiple viewports at different scales, the model space may have overlapping objects from the design layers, and the scale will be set to 1:1 instead of the expected design layer scale(s). To avoid this problem, set the scale the same for all design layers, and make sure that no objects overlap. Then, adjust the viewport scales as desired.
3. Do not export design layers in perspective, as they will not be in perspective in the DXF/DWG file. Instead, create a sheet layer with a viewport set to perspective view.
4. For DXF/DWG versions earlier than 14, object fills will not export. To better approximate the appearance of the exported file for these versions, remove all fills in a copy of the file before export. This will help identify lines under solid fills that need to be deleted or trimmed when the solid fill is removed.

   DXF/DWG versions 14 and later support associative boundary hatches, or “bhatches.” Therefore, fills, solid fills, and hatches can be exported as follows.

Vectorworks Entity	DXF/DWG Entity
Associative hatch definition	Select **Export Hatches** to create a bhatch definition; also select **Export Hatch Pattern Files** to export a hatch pattern file (.pat) along with the DXF/DWG file
Hatch with multiple colors or with background fill	Select **Export Hatches** to create multiple bhatch definitions; also select **Export Hatch Pattern Files** to export multiple hatch pattern files (.pat) along with the DXF/DWG file
Filled polygon with three or four sides, with the fill color different from the background color	Select **Export 2D Fills** to create a “solid” entity
Object with a solid fill (color different from background color) or hatch	Select **Export 2D Fills** to create an object with associated bhatch
Object with a solid fill (color same as background color)	Select **Export 2D Fills** to create one of the following: • Wipeout entity (AC2000 and later) • No fill (AC14 and earlier) • Light gray bhatch (AC14)
Filled polyline with one or more holes	Select **Export 2D Fills** or **Export Hatches** (depending on the Vectorworks fill) to create multiple objects and an associated bhatch definition with island detection

5. Set the class and layer visibility appropriately. If invisible Vectorworks layers are exported as DXF/DWG layers, objects in invisible classes will not be exported, and vice versa. To preview what will be exported, set classes to “show/snap/modify others” and layers to “show others.”

6. DXF/DWG does not support hybrid objects. If only design layers are being exported, set the view for each design layer so the appropriate 2D or 3D components are exported. From Plan view, the 2D component of a hybrid symbol is exported; from Top view or any 3D view, the 3D component is exported. To preserve the 3D object coordinates, use Top view to export all 3D design layers (or use viewports to show other views, and export the sheets containing those viewports). Ensure that the option **Export as Flattened 2D Graphics** is not selected if you want to export 3D coordinates.
7. Symbols, plug-in objects, and layer links export as named blocks in DXF/DWG. Groups can be exported as either named blocks or anonymous blocks.

   The name that these items receive depends on the view that is in effect upon export. As an example, the symbol "Widget" would receive the following names, depending on the view:

Symbol Type and View	Exported Block Name
Hybrid symbol in Plan view	Widget (2D)
Hybrid symbol in 3D view	Widget (3D)
Non-hybrid symbol	Widget

   The assigned layer link depends on whether **Project 2D** was selected for the layer link.

Layer Link Option	Exported Block Name
Layer link in plan view with **Project 2D** option selected	Layer-1 (2D)
All other layer links	Layer-1 (3D)

   To check the item names, re-import an exported file into a Vectorworks file.
8. Check the drawing with both black and white backgrounds. If the Vectorworks file has a white background and uses a dark (but not black) color to draw objects, then others who look at the exported file with a black background may not be able to see the objects.
9. Set the 2D and 3D conversion resolutions (in the Vectorworks preferences) to a low value if file size is a concern. Higher resolutions result in larger exported file sizes. Lower resolutions decrease file sizes, but can result in blockiness, especially when exporting to earlier versions of DXF/DWG.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

DXF/DWG and DWF Export Procedure
DXF/DWG and DWF Export Options
DXF/DWG and DWF File Formats

DXF/DWG and DWF Export Procedure

You can export a single DXF/DWG or DWF file, or export several files as a batch.

It is frequently helpful to view the translated file in a third-party software program to ensure that the translation process did not introduce unwanted effects. After export, leave the file open and import the exported file back into a Vectorworks file or into another software package. Compare the two versions, noting any problems. For example, the loss of object fills in earlier versions of DXF/DWG can cause unwanted lines to display. Correct these problems in a copy of the Vectorworks file and export the file again. Export options can be saved as a set.

Do not change the file extension of the exported file, or other software packages will not be able to read the file. Do not export as binary DXF unless you are certain that the recipient will be able to read this format.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

AutoCAD users may report that circles and other objects look like blocky polygons. They can improve the appearance of the drawing by zooming in and using either the “regenall” or “regen” command, or by increasing the VIEWRES value.

Exporting a Single DXF/DWG or DWF File

1. Select **File > Export > Export DXF/DWG** or **File > Export > Export DWF**.

   Either the DXF DWG Export Options or the DWF Export Options dialog box opens, depending on your selection. The dialog boxes are very similar, except for the file format settings.

2. Select the appropriate export options, and click **OK** to export the file.

   The export dialog box options are described in “DXF/DWG and DWF Export Options” on page 1734. During the export, a progress bar displays, along with the number of objects processed.

Exporting DXF/DWG or DWF Files as a Batch

Use the **Publish** command to export a series of sheet layers and/or saved views from the current drawing and externally-referenced files to DXF/DWG or DWF format. See “Batch Publishing” on page 1765.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

DXF/DWG and DWF Export Options
Batch Publishing

DXF/DWG and DWF Export Options

The DXF/DWG Export Options dialog box and the DWF Export Options dialog box are very similar, except for the file format settings. This is because when you export to DWF, the drawing objects are first converted to DXF/DWG, and then output in DWF format. Change the export options as needed to retain the integrity of the Vectorworks drawings in the DXF/DWG format. The same export options display for export of single files and batches of files.

To make custom exports faster and easier, save the sets of options that you use frequently.

Click to show/hide the parameters.

Parameter	Description
Saved Settings Options	
Settings	Select <Active Settings> to use the export options currently selected in the dialog box; to use a set of saved export options, select them from the list
Save	Opens a dialog box to name and save the currently selected export options so that they can quickly be selected as a set
Manage	Opens the Saved Settings dialog box to rename or delete sets of saved export options (see “Managing Saved Export Options” on page 1741)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
File Format	
Format	DXF/DWG export: • Choose from three exported file formats: DXF as plain text, DXF as a binary encoding, and DWG. • One of the main differences between these formats is file size. Although individual files can vary, an uncompressed text DXF file is generally larger than an uncompressed DWG file. However, the file size can be significantly reduced by using a compression utility. A compressed text DXF file is normally smaller than a compressed DWG file. Binary DXF file sizes generally fall between the two for both compressed and uncompressed files. • In addition to file size, consider the formats that other applications can support. Before exporting to binary DXF or to DWG, ensure that the receiving party's software can read those formats. Since binary and text DXF both use the same .dxf extension, if the DXF file cannot be opened, it may be may be mistakenly considered corrupted. When in doubt, export as text DXF, which is universally supported. • Recent versions of AutoCAD have bugs when reading some DXF files, such as losing links to images, so DWG is usually a safer choice in that situation. • DXB is a simpler file format used by some third-party applications that do not support the full DXF or DWG file format. It is not the same as binary DXF; do not use .dxb as the extension for binary DXF files. The Vectorworks program does not support DXB. DWF export: • Choose from five file formats: DWF as plain text, DWF as a regular or compressed binary encoding, 3D DWF, or DWFx.
Version	For best results, export to the highest version supported by the recipient's software, or the default export version, whichever is lower. The latest versions of DXF/DWG and DWF have features more similar to the Vectorworks program and may provide a better translation, but not all software packages can read the latest versions. When in doubt, export as DXF and version 12, which has widespread industry support. Other options on the dialog box can change or become unavailable depending on the version selected.
Class/Layer Conversions	
Export as DXF/DWG Layers	This option is available if the **Export** option is set to Design Layers Only. Since Vectorworks classes correspond most closely to DXF/DWG layers, normally, the **Classes** option is recommended. There is no direct equivalent for Vectorworks layers in DXF/DWG. A single Vectorworks design layer is similar to DXF "model space." This is most important for the import of groups and symbols. In a Vectorworks file, the objects in groups and symbols can belong to different classes, but they must be on the same layer. In a DXF/DWG file, the entities equivalent to Vectorworks symbols and groups (known as "blocks") can be on different DXF/DWG layers. Layers normally do not export relative to the **Elevation** value shown in the Design Layers tab of the Organization dialog box.

Parameter	Description
Invisible Classes/ Layers Are	DXF/DWG export: While a Vectorworks file organizes drawings by class and by layer, the DXF/DWG file format only has layers. The following description assumes that, as recommended, classes are selected for export as layers. (If instead Vectorworks layers are selected for export as DXF layers, equivalent options are presented for the layers.) Select whether to export invisible classes or layers. If **Exported As Invisible DXF/ DWG Layers** is selected, objects that are in invisible classes in the Vectorworks file are exported, and can be seen by making the DXF/DWG layer visible. This is the recommended option. However, if invisible classes contain private information or if the size of the exported file needs to be reduced, select **Not Exported** to delete these. If invisible classes are exported as DXF layers, objects on invisible Vectorworks layers are not exported. To export these items, first make the layers visible, and then select the **Export** command. DWF export: This option is unavailable; only visible graphics can be exported to DWF.
Export Layers as Separate Files	• For single file export, select this option to export the selected sheet layers or design layers to separate DXF/DWG or DWF files (named after the original layer); otherwise, the selected layers are exported to one file. This option is unavailable if the **Export** option is set to Design Layers Only, and **Export As DXF/DWG Layers** is set to Layers. Additionally, DWF versions 4.2 and 5.5 do not support multiple sheets, so this option is unavailable if multiple sheet layers are selected for export to one of these versions. All design layers are exported, including those that are set to be invisible. • For batch file export, this setting is ignored; each sheet layer is always exported as a separate file.
Layer Scale	
Rescale Layers To	When the **Export** option is set to Design Layers Only, paper space is not used; all items are placed in model space. Model space must be at one scale; DXF/DWG files do not have different layer scales. If the layers to be exported are at various scales, the option to rescale them to a common scale before export becomes available. By default, the most frequently used layer scale will be used. Choosing the best scale for export is important. Select **Rescale Layers To** and click the common scale to use from the displayed list. Symbols on rescaled layers are exported as scaled blocks in the DXF/DWG file. When sheets are selected for export, all design layers export to model space at effectively a 1:1 scale, and viewports take care of showing the objects at other scales.

Parameter	Description
Sheets to Include	
Export	The options on this list change depending on the contents of the file being exported and on which export **Version** is selected. Select the items to export from the list. By default, the selected layers are exported to a single file (to export multiple files instead, select **Export Layers As Separate Files**). • If Design Layers Only is selected, you can select either classes or layers to **Export as DXF/DWG Layers**. If you export the design layers as DXF layers, you cannot select **Export Layers As Separate Files**. Additionally, you have the option to **Export as Flattened 2D Graphics** when you export design layers only. • If one of the sheet options is selected, the selected sheet layers are exported as paper space layouts; also, design layers used in viewports are exported to model space. While multiple sheets can be exported to the same file, note that unrelated sheets usually should be exported to different files. Select the sheets from the list below. • If one of the saved view options is selected, saved views are each exported as separate files. Select the views from the list below.
Export Viewports as 2D Graphics in Model Space	If one of the sheet options is selected, select this option to export all 2D and 3D objects that display inside a viewport as 2D projected (flattened) graphics in model space. Each selected sheet is exported to a separate file. On re-import, all exported graphics are imported into a design layer. • Rendered viewports in any view other than Top/Plan view are exported as groups. • Non-rendered viewports in an orthogonal view (Top, Bottom, Front, Back, Left, and Right) are set to hidden line rendering and then exported as groups. • Non-rendered viewports in an isometric view (including custom views) are converted to lines before export. • Section viewports are exported as groups. • Viewport annotations are extracted from the viewport groups and exported as single entities. • All other sheet layer graphics (such as title blocks) are exported as single entities.
Sheets or View Name	If Selected Sheets or Selected Saved Views is selected as the **Export** option, select the sheets or saved views to export from the displayed list.
References	
Export Design Layer Viewports as Separate Files (Vectorworks Design Series required)	Select this option to export each design layer viewport as a separate DXF/DWG file, named with the viewport name. If deselected, design layer viewports are bound into the master DXF/DWG file; this may cause the viewport objects to look different, because class and layer visibility overrides are not retained. This option is available only for DXF/DWG export.
Objects	
Export Only Selected Objects	Select this option to export only the objects that are currently selected in the drawing; this option is unavailable if no objects are selected
Export as Flattened 2D Graphics	If you are exporting design layers only, select this option to export all of the objects from the current design layer as flattened 2D graphics

Parameter	Description
Text	
Preserve Mapped Font on Export	If the file contains text that has been mapped to a different font, select this option to export the text with the replacement font; if the option is disabled, the text is exported with the original font
2D Fills and Files	
Export 2D Fills	Select this option to export solid fills as wipeouts (DXF/DWG versions 2000 and later) or solid hatches (DXF/DWG version 14 and later). If **Export Images and Image Files** is enabled, image fills export as clipped images; if the option is disabled, image fills export as solid fills.
Export Images and Image Files	Select this option to export image objects and image files. If this option is disabled, objects with image fills export as objects with solid fills. This option is only enabled for DXF/DWG versions 2000 or higher.
Export Hatches	Select this option to export Vectorworks hatches as bhatches; no hatch pattern files are created unless **Export Hatch Pattern Files** is also selected
Export Hatch Pattern Files	If **Export Hatches** is enabled, select this option to create additional hatch pattern (.pat) files in a specified folder. A Vectorworks hatch with multiple levels and colors generates multiple hatch pattern definitions. See “Preparing to Export” on page 1732 for more information on the conversion. The hatch pattern files, and the DXF/DWG exported file and any support files, such as .jpg images, are placed in this folder. AutoCAD requires the hatch pattern files to retain the hatch associativity and to add hatches to additional objects with the same hatch pattern. When **Export Hatch Pattern Files** is disabled, AutoCAD displays the hatch but cannot edit it. This option is unavailable during DWF export.
Export Hatches and 2D Fills into Separate DXF/DWG Layers	Select this option to export all of the fills, patterns, gradients, and hatches within a class (or design layer, if layers are exported to DXF/DWG layers) to a separate DXF/DWG layer. The DXF/DWG layers are named after the original class or layer that contains the object to which the fill, pattern, gradient, or hatch is applied. For example, if the Vectorworks file has an object on the “Hardscape” layer with a hatch applied to it, the export file will have a layer called “Hardscape_Hatch.”
3D	
Export Solids as ACIS Solids	Select this option to export most Vectorworks solids and NURBS surfaces as ACIS objects. Deselect this option to export solids as polygonal faces (if the target software package cannot read ACIS solids, for example). Walls, round walls, roof and floor slabs, filled 3D polygons, meshes, and NURBS curves cannot be exported as ACIS solids. This option is unavailable during DWF export.

Parameter	Description
Triangulate to Preserve Fills	Some software packages, such as AutoCAD, are not capable of rendering exported 3D surfaces that have more than three or four vertices per face. Select this option to break up such faces into a set of triangles that can be properly rendered. The algorithm used works best on planar or nearly planar surfaces, such as the top of an extruded circle. Even if **Export Solids as ACIS Solids** is selected, this option may be used to handle 3D objects that cannot be exported as ACIS solids. When in doubt, select this option (this increases the exported file size and the time necessary to export it, and may result in unwanted lines in some cases).
Symbols and Groups	
Decompose 3D Symbols and Groups	Some software packages cannot handle exported groups and symbols. If a problem occurs, select **Decompose 3D Symbols and Groups** to convert symbols and groups to ungrouped objects. Do not select this option unless it is absolutely necessary. This option is unavailable during DWF export.
Export Groups as Anonymous Blocks	Normally, Vectorworks groups are exported as named blocks, which can be opened and edited in other software packages. However, the named blocks are imported back into Vectorworks files as symbols. If this creates a problem, select **Export Groups as Anonymous Blocks**, which are imported back into Vectorworks files as groups. This option is unavailable during DWF export.
Dimensions	
Preserve SIA Dimension Text Appearance	Select this option to maintain the SIA dimension superscript formatting upon export. Dimensions are converted to non-interactive objects. Deselect this option to change the SIA dimensions to a non-superscript dimension format.
Line Types	
Export Complex Line Types as Blocks	Select this option to export complex Vectorworks line types as anonymous blocks, which preserves the exact appearance of the drawing. If the option is disabled, complex line types export as true line types, and additional graphic information is compiled into a shape file (.shx).
Line Weights and Colors	
Use True Colors	Exports line colors with the Vectorworks RGB values, and converts line weights to the closest possible DXF/DWG weight value. Named Vectorworks colors export as named DXF/DWG true colors. • For DXF/DWG versions prior to version 2000, line weights are not supported, and this option should not be selected; select **Use DXF/DWG Indexed Colors** and **Map Line Weights to Colors** instead. • For DXF/DWG version 2000 and later, select this option only if the exported file will not be used with a .ctb file for plotting or printing in AutoCAD. If the exported file is intended to be used with a .ctb file later on, select **Use DXF/DWG Indexed Colors** instead.

Parameter	Description
Use DXF/DWG Indexed Colors	Exports line colors with DXF/DWG indexed color values, and converts line weights to the closest possible DXF/DWG weight value. Unlike **Use True Colors**, this option allows you to use the exported file with a .ctb file for plotting or printing in AutoCAD. You can either use a predefined .ctb file, or select **Map Line Weights to Colors** to create a .ctb file specific to this drawing upon export.
Map Line Weights to Colors	If **Use DXF/DWG Indexed Colors** is selected, this additional option creates a .ctb file for each exported DXF/DWG file (named after the original .vwx file) to be used during plotting or printing in AutoCAD. The AutoCAD user must put this file in the support path, as AutoCAD does not read the .ctb file when it is simply included in the same folder as the DXF/DWG file. During the export process, a list of line weights in the file displays. Specify the color to map to each of these line weights. To select a different color for a selected line weight, click the color box and select a color from the displayed options. If **Export Layers As Separate Files** is also selected, the mapping dialog box does not display; instead, the Vectorworks program automatically maps line weights to colors according to a predefined standard. To avoid having a color table file for each exported file, use a single .ctb file for each unique set of mappings, and delete the others. When an exported file is opened in AutoCAD, edit the page setup and choose an appropriate color table file. When you import the file back into the Vectorworks program, use the reverse process to convert the colors back to line weights.

Line Weight Conversions

VW Line Weight (mm)	DXF Line Weight (mm)
0	Deleted
0.01 - 0.07	0.05
0.08 - 0.11	0.09
0.12 - 0.14	0.13
0.15 - 0.16	0.15
0.17 - 0.19	0.18
0.20 - 0.22	0.20
0.23 - 0.27	0.25
0.28 - 0.32	0.30
0.33 - 0.37	0.35
0.38 - 0.45	0.40
0.46 - 0.51	0.50

VW Line Weight (mm)	DXF Line Weight (mm)
0.52 - 0.56	0.53
0.57 - 0.65	0.60
0.66 - 0.75	0.70
0.76 - 0.85	0.80
0.86 - 0.95	0.90
0.96 - 1.03	1.00
1.04 - 1.13	1.06
1.14 - 1.30	1.20
1.31 - 1.49	1.40
1.50 - 1.75	1.58
1.76 - 2.05	2.00
2.06 - 6.48	2.11

Managing Saved Export Options

If you have saved a set of export options, use the **Manage** button on the DXF DWG Export Options (or DWF Export Options) dialog box to rename or delete these saved settings when needed.

To manage saved DXF/DWG or DWF export settings:

1. Select **File > Export > Export DXF/DWG** (or **Export DWF**). The appropriate export options dialog box opens.
2. Click **Manage**.

   The Saved Settings dialog box opens. Select the set of saved export options to change.

Click to show/hide the parameters.

Parameter	Description
Rename	Select and enter a new name for the set of options. If the entered name is already assigned to another set of options, you are prompted to confirm that you want to replace the existing set with the set being saved.
Delete	Select to delete the set of options. You are prompted to confirm that you want to delete the saved settings.

3. Click **OK** to close the Saved Settings dialog box. Click **OK** again to close the export options dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

DXF/DWG and DWF Export Options

Items Which Cannot Export to DXF/DWG or DWF

Certain Vectorworks items have no equivalent in DXF/DWG or DWF and therefore cannot be exported. Other items could be exported, but inadequate support in AutoCAD makes it inadvisable to do so. A few objects (such as worksheets) could in theory be exported to something useful, but the Vectorworks program currently lacks support for doing so.

The following list includes items that do not export to DXF/DWG or DWF, for various reasons.

- Textures
- Lights
- Image props
- Individual layer scales
- EPS or vector PICTs in some cases
- “Graying” of layers and classes
- Renderworks background images
- Record format information attached to arbitrary objects
- Design layers with perspective views
- Oblique cavalier and oblique cabinet projections
- Object names attached to arbitrary objects
- Layer transfer modes other than **Paint**
- Depending on the export options, either layers or classes
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

IFC Format Interoperability

A L IFC Overview

The Vectorworks Architect and Landmark products support Building Information Model (BIM) interoperability using the Industry Foundation Classes (IFC) data specification and file formats. Vectorworks currently supports versions 2x2 and 2x3 and is certified by buildingSMART International for the export of models based on the IFC2x3 Coordination View 2.0 – Architecture model view definition, as well as import of any IFC2x3 Coordination View 2.0 model.

IFC is an open standard, developed and maintained by buildingSMART International, for building data that permits information to be shared and maintained throughout the life cycle of the construction project: design, analysis, specification, fabrication, construction, and occupancy.

The capabilities and usage for IFC data standards are still evolving. However, objects and models containing IFC data have been proven to be a useful part of the design process. Some examples of the use of IFC data include:

- Collaborative design, where an architect exports an architectural model to a structural engineer for analysis and design, and then imports the structural model, in an IFC format, for coordination purposes
- Energy performance simulation and analysis of a building envelope and systems
- Automated analysis of code compliance
- Space planning and space inventory analysis

Central to the concept of IFC is the idea of "semantic objects." Like Vectorworks plug-in objects, IFC objects are more than just collections of geometry; they have a meaning within the building fabric itself, be it as a door, a wall, a window, or a handrail. The following Vectorworks plug-in objects have default IFC2x3 object types and properties assigned, but they may be edited by the user as necessary. These Vectorworks plug-in objects are automatically converted upon export to the corresponding IFC object types shown. Additionally, most content in the Vectorworks Architect, Landmark, and Designer symbol libraries has default IFC2x3 object types and properties assigned, but symbols may be edited by the user, as necessary, and are automatically converted upon export.

Base Cabinet [IfcFurnishingElement]	Bath-Shower [IfcFlowTerminal]
Ceiling Grid [IfcCovering]	Clothes Rod [IfcFurnishingElement]
Column [IfcColumn]	Comm Device [IfcDistributionFlowElement]
Compartment Sink [IfcFlowTerminal]	Counter Top [IfcFurnishingElement]
Curtain Wall (Straight & Curved) [IfcCurtainWall]	Curtain Wall Frames [ifcMember]
Curtain Wall Panels [IfcPlate]	Desk [IfcFurnishingElement]
Door [IfcDoor]	Drilled Footing [IfcFooting]
Simple Elevator [IfcTransportElement]	Escalator [IfcTransportElement]
Fireplace [IfcDistributionFlowElement]	Floor [IfcSlab]
Framing Member [IfcMember or IfcBeam]	Grab Bars [IfcRailing]
Guardrail (Curved & Straight) [IfcRailing]	Handrail (Curved & Straight) [IfcRailing]
HVAC Damper [IfcDistributionFlowElement]	HVAC Diffuser [IfcDistributionFlowElement]
HVAC Elbow Duct [IfcDistributionFlowElement]	HVAC Flex Duct [IfcDistributionFlowElement]
HVAC Outlet [IfcDistributionFlowElement]	HVAC Splitter [IfcDistributionFlowElement]
HVAC Straight Duct [IfcDistributionFlowElement]	HVAC Transition [IfcDistributionFlowElement]
HVAC Vertical Duct [IfcDistributionFlowElement]	HVAC Vertical Elbow [IfcDistributionFlowElement]

Incandescent Fixture [IfcDistributionFlowElement]
Landscape Wall (including Arc and Bezier) [IfcWall]
Massing Model (on layer mapped to site) [IfcBuilding]
Mullion [IfcMember]
Parking Spaces [IfcSpace]
Pilaster [IfcColumn]
Pillar [IfcColumn]
Piping Run [IfcDistributionFlowElement]
Plant [IfcBuildingElementProxy]
Ramp [IfcRamp]
Receptacle [IfcDistributionFlowElement]
Roadway (all types) [IfcTransportElement]
Roof [IfcRoof containing instances of IfcSlab]
Roof Face [IfcSlab]
Seating Layout [IfcFurnishingElement]
Round Wall [IfcWall or IfcWallStandardCase]
Wall [IfcWall or IfcWallStandardCase]
Site Model [IfcSite]
Shelving Unit [IfcFurnishingElement]
Slab [IfcSlab]
Space [IfcSpace]
Stair (including Simple and Custom) [IfcStair]
Switch [IfcDistributionFlowElement]
Table [IfcFurnishingElement]
Tables and Chairs [IfcFurnishingElement]
Toilet Fixture [IfcFlowTerminal]
Utility Cabinet [IfcFurnishing Element]
Wall Cabinet [IfcFurnishingElement]
Window [IfcWindow]
Workstation Counter [IfcFurnishingElement]
Workstation Overhead [IfcFurnishingElement]
Workstation Panel [IfcFurnishingElement]
Workstation Pedestal [IfcFurnishingElement]

With the **IFC Data** command, users can assign object types and properties to arbitrary collections of geometry so that other applications that use these IFC files can identify these objects.

When an IFC file is imported into a Vectorworks file, IfcSpace objects are translated into the corresponding Vectorworks Space object. Since the properties of other objects cannot easily be matched to all the corresponding Vectorworks object properties and controlling parameters, the objects are imported into the Vectorworks file as a particular and flexible kind of plug-in object: an “IFC Entity.” The semantic definition of the IFC Entity—IfcColumn, IfcWall, IfcWindow, for example—is maintained and displayed. Like a group or symbol, an IFC Entity can contain an arbitrary collection of geometry, usually generic or constructive solid (CSG), but like a plug-in object, it can be inserted into walls and has sets of data and properties attached. With this generalized import capability, the Vectorworks program can import any kind of object supported by the IFC data standard, regardless of whether it has a corresponding parametric plug-in object.

Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Assigning IFC Data to Objects
Assigning IFC Data to Space Zones
Viewing and Editing IFC Data
Using Custom IFC Property Sets
Creating IFC Schedules
Importing IFC Files
Exporting IFC Projects
Viewing BCF Files
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

IFC Workflows

Generally, IFC-compatible applications are set up to work with an entire project (sites with buildings consisting of stories) in a single file. When you use the **Export IFC Project** command, you assign design layers to building stories for meaningful export (Vectorworks Architect required). For a project initially set up with stories, mapping is done automatically so that the layers assigned to the building stories are automatically included in the Mapped Layers list and are mapped to an appropriately named IFC Story. Mapping can also be controlled manually, allowing you to override or edit the automated process or make any subsequent changes. The **Import IFC** command automatically assigns IFC building stories to their own design layers.

To make Vectorworks building models easy to export to IFC, the following guidelines may be useful:

- Set up the project initially with stories as described in “Setting Up the Building Structure with Stories” on page 172 (Vectorworks Architect required).
- Design layers for the building model should correspond to building stories (Vectorworks Architect required), not categories of information. (Use classes for categorization.)
- Take care to set up the elevation values of layers correctly, and verify that all the objects in the drawing are aligned correctly vertically. Examining the project in Unified View mode makes this much easier.
- Use standard Vectorworks plug-in objects listed in “IFC Format Interoperability” on page 1743 wherever possible.
- Use the **IFC Data** command to attach IFC data to custom user-defined hybrid or 3D symbols, custom 3D elements, or auto-hybrid objects, so that they will be recognized at IFC export.
- Use the **VA Create Schedule** command (Vectorworks Architect required) to generate default IFC schedules, or create your own schedules to verify and correct the IFC data attached to objects. See “Records and Schedules” on page 1859.
- Test the quality of the IFC export file by verifying it in an IFC model viewer. Almost all IFC viewers can traverse the IFC hierarchical “tree” structure to view and validate the model. Some examples of IFC viewers available at this time include:
  - Solibri Model Viewer (Macintosh or Windows)
  - FZKViewer (Windows)
  - DDS-CAD Viewer (Windows)
  - Tekla BIMsight (Windows)
  - Constructivity Model Viewer (Windows)
  - RDF IFC Viewer (Windows; Macintosh and Linux versions planned)
  - Field 3D (iOS)
  - IFC WebServer (web browser)
  - xBIM Xplorer (Windows)
  - IfcPlusPlus (Windows)

  For more information about additional available IFC viewers and other IFC-compatible applications, visit the buildingSMART IFC-Compatible Implementations Database website or IfcWiki.org.
- If you receive comments or corrections for the project in BIM Collaboration Format (BCF), use the **Open BCF Viewer** command to view the notes, and then correct the project file as needed.

~~~~~~~~~~~~~~~~~~~~~~~~~~

Assigning IFC Data to Objects
Assigning IFC Data to Space Zones
Viewing and Editing IFC Data
Creating IFC Schedules
Importing IFC Files
Exporting IFC Projects
~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Up the Building Structure with Stories
Viewing BCF Files

Assigning IFC Data to Objects

When a building project is exported to an IFC file, Vectorworks plug-in objects and pre-assigned symbols are automatically assigned to IFC entities. Simple hybrid or 3D geometric objects, unless they receive IFC assignments, are not exported. To be exported, these object associations must be made prior to export with the **IFC Data** command. Assigning IFC data to an object does not alter it in any visible way. The object can still be edited with standard Vectorworks tools and commands. An example of this would be to use a floor or slab object to model a flat ceiling. Select the floor or slab in the model, and then select the **IFC Data** command. From the list in the Select IFC Entity dialog box, select IfcCovering, the correct assignment for a ceiling. Then, in the IFC Data dialog box, select CEILING from the PredefinedType list in the **Properties for the selected Data Set** lists.

The **IFC Zones** command attaches IFC data to space zones. See “Assigning IFC Data to Space Zones” on page 1748.

An object can also be “wrapped” in a container so that it is identified in the Vectorworks file as an IFC Entity object. Editing an IFC Entity is less direct than editing simple Vectorworks objects with IFC data attached. For this reason, an IFC Entity can be useful for model elements that need to be secured from casual changes.

To assign IFC data:

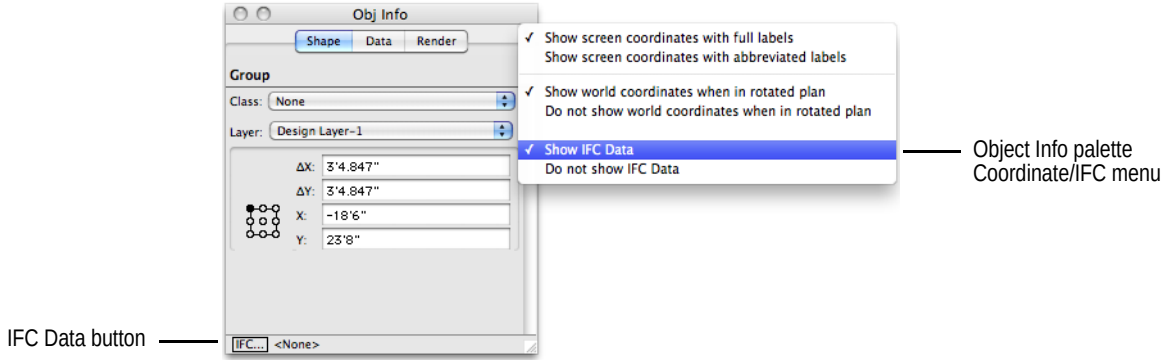
1. Select the object, group, symbol instance, or symbol definition for assignment of IFC data. More than one item can be selected at one time.

   IFC data applied to symbol instances affect only the selected instances, not all instances. To attach IFC data to symbol definitions, select the symbol definition in the Resource Browser. Right-click (Windows) or Ctrl-click (Macintosh) on the symbol and select **IFC Data** from the context menu. This attaches the IFC data to future symbol placements, and to any current instances that do not have IFC data already attached to them.

2. Select the **IFC Data** command from the appropriate menu:

   - Architect workspace: **AEC > IFC Data**
   - Landmark workspace: **Landmark > Architectural > IFC Data**

   Alternatively, select **Show IFC Data** from the **Coordinate/IFC** menu on the Object Info palette to enable IFC-related information display on the palette, and then click **IFC Data** from the Object Info palette.



   The Select IFC Entity dialog box opens, listing available IFC object types.

Click to show/hide the parameters.

Parameter	Description
Simple List/Full List view	Choose the list view. The simple list is an adjusted view of primary IFC objects, while the full list displays all available IFC concepts
Filter List	Enter text or keywords to filter the objects list; only entries which contain the text are listed
IFC object types	Lists available IFC object types; click the object type to select it

3. Select the object type and click **OK**.

   The IFC Data dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Entity	Displays the selected type of IFC entity
Select	Opens the Select IFC Entity dialog box, to select a different entity type
GUID	Displays the Global Unique Identifier assigned to the entity; this ID follows the entity throughout the IFC workflow to distinguish it from other IFC objects
Create IFC Entity	Leave deselected to retain the Vectorworks object properties in the Object Info palette (the IFC Data remains attached); select to display the object as an IFC Entity in the Object Info palette. IFC Entities are container objects. They display the type IFC Entity on the Object Info palette.
Data Sets for this object	Displays the property sets associated with the IFC object; select whether to use each IFC pSet by clicking in the **Use** column
Manage Custom Property Sets	Opens the Manage Custom Property Sets dialog box. This allows you to include data from your model even if the data are not defined as part of the IFC specifications.
Properties for the selected Data Set	Displays the available properties and values for the selected pSet
Value for selected property	Specifies a string or number value for the selected pSet property
Select IFC Instance	For some pSet properties with complex values, an IFC Instance is specified. Select from the list of instances, or click **New** to open the IFC Instance Edit dialog box. Specify the values of the complex properties and click **OK** to return to the IFC Data dialog box. (To delete an IFC instance, select it from the list and click **Delete**.)

4. Click **OK** to assign the IFC data to the object, or create the IFC entity. The Object Info palette displays the selected IFC object or entity information (IFC value type and the object name, if any) when **Show IFC Data** is enabled on the palette. Objects like walls and slabs, with default IFC data assigned, display with <Default> to indicate that the default IFC data have not been modified.

The geometry of IFC entities can be edited with the **Modify > Edit IFC Entity** command; alternatively, right-click (Windows) or Ctrl-click (Macintosh) on the IFC entity and select **Edit** from the context menu.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Assigning IFC Data to Space Zones
Viewing and Editing IFC Data
IFC Format Interoperability
Using Custom IFC Property Sets
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Importing IFC Files
Exporting IFC Projects

Assigning IFC Data to Space Zones

The **IFC Zones** command attaches IFC data to the specified IFC zones from the current file.

Zones are assigned to space objects from the Occupancy pane of the Space Settings dialog box (see "Space Settings: Occupancy Pane" on page 408).

To attach IFC data to zones in use in the file:

1. Select the **IFC Zones** command from the appropriate menu:
   - Architect workspace: **AEC > IFC Zones**
   - Landmark workspace: **Landmark > Architectural > IFC Zones**

   The IFC Zones dialog box opens. Zones in use in the file are listed above the divider line; available IFC zones are listed below the divider.
2. Select a zone and click **OK**. Only one zone can be selected at a time.
3. The IFC Data dialog box opens. The object type is automatically set to ifcZone, and the property Name value set to the zone name. See "Assigning IFC Data to Objects" on page 1746.
4. Click **OK** to attach the IFC data to the zone.

   At IFC export, the IFC zone data is automatically applied to all space objects with a matching zone definition.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Assigning IFC Data to Objects
Viewing and Editing IFC Data
IFC Format Interoperability
Using Custom IFC Property Sets
Importing IFC Files
Exporting IFC Projects

Viewing and Editing IFC Data

The data attached to custom Vectorworks objects or IFC entities, whether created in the Vectorworks program or imported from an IFC file, can be viewed and edited with the **IFC Data** command or from the Object Info palette. In addition, the IFC object that corresponds to a Vectorworks plug-in object, such as a Space object, window, or wall, can be viewed.

To access IFC data:

1. Select the IFC entity or Vectorworks plug-in object. Multiple items can be selected.
2. Select the **IFC Data** command from the appropriate menu:
   - Architect workspace: **AEC > IFC Data**
   - Landmark workspace: **Landmark > Architectural > IFC Data**

   Alternatively, click **IFC Data** from the Object Info palette (**Show IFC Data** must be enabled from the **Coordinate/IFC** menu on the Object Info palette to enable IFC-related information display on the palette).

   When multiple items have been selected, edits apply to all eligible objects in the selection.

   If you have selected a group that contains objects with multiple IfcObjectTypes, the Choose IFC Type dialog box opens. Select the IFC Type to apply to the group. Optionally remove the IFC data from the contained objects.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

3. When the items to be edited have been specified if necessary, the IFC Data dialog box opens. View or edit the IFC data as described in "Assigning IFC Data to Objects" on page 1746 and click **OK**.

   If an IFC entity was selected, the IFC Data dialog box displays the data attached to the entity as described in "Assigning IFC Data to Objects" on page 1746; the data can be edited. If a Vectorworks plug-in object was selected, the IFC Data dialog box displays the corresponding IFC Object type. At export, the object will be converted to that type of entity.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

IFC Format Interoperability
Using Custom IFC Property Sets
Importing IFC Files
Exporting IFC Projects

A L Using Custom IFC Property Sets

Property sets (Pset_XxxxXxxx) are used to assign groups of individual data fields, or properties, to IFC entities. Property sets can be very specific to an IFC entity, such as Pset_WallCommon to IfcWall, or more generally applied to any IFC entity, such as Pset_ManufacturerTypeInformation. Some property sets and their values are explicitly predefined and listed in the IFC specification; the naming convention Pset_Xxx applies to these officially specified property sets.

Due to the extensible nature of IFC, any user-defined data, even when not explicitly identified in the IFC specification, can still be captured and exchanged using custom property sets. Custom property sets must follow technical encoding conventions. They must have unique and allowable names; the names cannot duplicate specified property set names or begin with "Pset_."

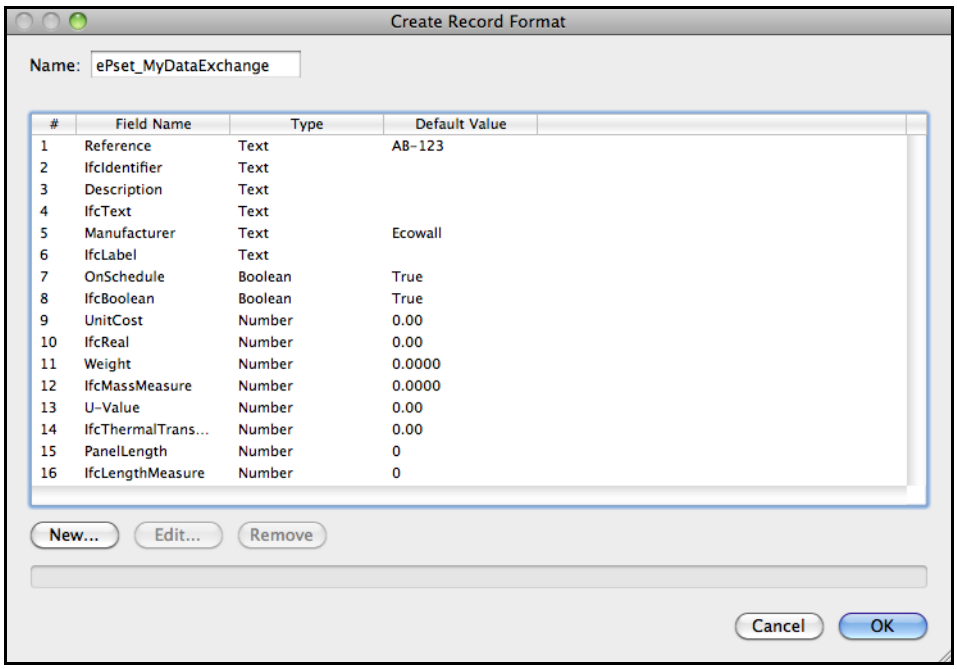
Record formats allow custom property sets to be saved and exchanged. See "Record Formats" on page 262 for more information on record formats.

Creating IFC-Compatible Record Formats

To use a custom property set, first create the record format to define the data to be captured and exchanged in IFC format. The record format name defines the name of the custom property set. Record formats designated for IFC export should be named with a VwPset_ or ePset_ prefix, or use a name that is defined by a specific Model View Definition (MVD) or documented model Exchange Requirement (ER). Record format names are case sensitive, and should not contain blank spaces (use an underscore if needed). In the following example, the record format is named "ePset_MyDataExchange."

The record format can consist of any number of fields, designated by a **Field Name** and a **Field Type**; see "Creating Record Formats" on page 262. To be compatible with IFC standards, each field name must be paired with a second field name that identifies an IFC value (IfcValue) type. Each pair must have the same field type.

For example, the **Field Name** "Reference," with a Text **Type**, is followed by the **Field Name** "IfcIdentifier," also with a Text **Type**. The "IfcIdentifier" field indicates the kind of value type being used; in this case, it is a simple text ID string (IfcSimpleValue:IfcIdentifier).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



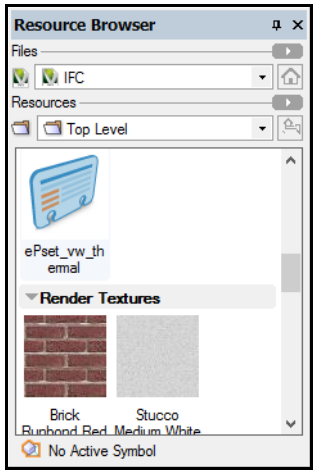
When creating record format fields, a default value can be specified. The default values are optional, except for number fields, which require at least a zero value to be entered. However, the default values for the IFC field names are ignored when converted to custom property sets.

Deciding which IFC value type and record format field type to use depends on the kind of information being captured by the field, such as a simple number, a TRUE/FALSE choice, a text string or simple label, or a measurement. In the Vectorworks program, allowable types include Integer, Boolean, Text, or Number. The **Field Name** identifying an IFC value type should be based on the IFC specification for the different defined types of values (IfcValue), as shown in the following list.

IFC Value Types	Vectorworks Record Format Field Type and Description
IfcSimpleValue	These are the most common and cover most user cases
IfcInteger	**Integer**: a simple whole number ranging from -32,7568 to 32,767
IfcReal	**Number**: General or Decimal
IfcBoolean	**Boolean**: also known as TRUE or FALSE
IfcLogical	**Boolean**: similar to Boolean, but can include a value of "UNKNOWN"
IfcIdentifier	**Text**: a simple text ID string, usually a mix of alphanumeric characters and symbols
IfcLabel	**Text**: a simple text name string, usually a mix of alphanumeric characters
IfcText	**Text**: a descriptive text field string of up to 255 characters
IfcMeasureValue	A complete list can be found in the IFC2x3 TC1 specification
IfcAreaMeasure	**Number**: Dimension Area
IfcLengthMeasure	**Number**: Dimension
IfcMassMeasure	**Number**: General or Decimal

IFC Value Types	Vectorworks Record Format Field Type and Description
IfcThermodynamicTemperatureMeasure	**Number**: General or Decimal
IfcTimeMeasure	**Integer** or **Number**: General
IfcVolumeMeasure	**Number**: Dimension Volume
IfcDerivedMeasureValue	A complete list can be found in the IFC2x3 TC1 specification
IfcEnergyMeasure	**Number**: General or Decimal
IfcIlluminanceMeasure	**Number**: General or Decimal
IfcPowerMeasure	**Number**: General or Decimal
IfcThermalTransmittanceMeasure	**Number**: General or Decimal
IfcTimeStamp	**Integer** or **Number**: General
IfcLuminousIntensityDistributionMeasure	**Number**: General or Decimal

When all field data pairs have been completed, click **OK** to create the record format.



The record format appears in the Resource Browser, under the Record Format category. Like any resource, it can be shared between files using the export or import functionality of the Resource Browser.

Converting a Record Format to an IFC Custom Property Set

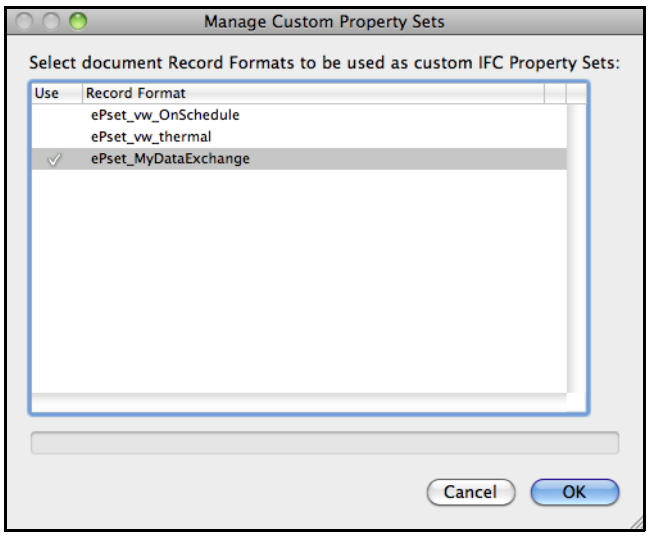
Once the custom record format has been created, it is converted to an IFC Property Set and attached to an object, group, symbol definition, or symbol instance.

To assign the custom IFC data:

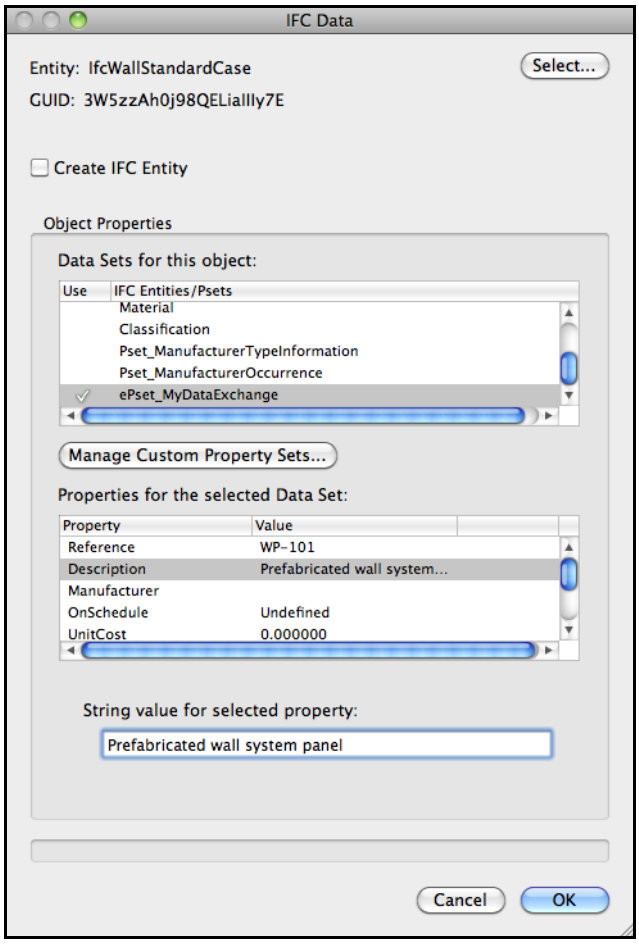
1. Select the object, group, or symbol for assignment of IFC data.
2. Select the IFC object type as described in “Assigning IFC Data to Objects” on page 1746.
3. Click **OK**.

   The IFC Data dialog box opens.
4. Click **Manage Custom Property Sets**.

   The Manage Custom Property Sets dialog box opens.



5. Click in the **Use** column to include a record format. Selected record formats display with a check mark.
6. Click **OK** to return to the IFC Data dialog box.



The selected custom record formats are listed as data sets in the **IFC Entities/Psets** list.

If there are any errors in the naming or formatting of any fields in the record format, the custom property set does not appear in the **IFC Entities/Pset** list. Internal checks ensure the integrity of the data and automatically reject any errors. Correct the errors by editing the record format from the Resource Browser, and try again.

7. Select the custom Pset and click in the **Use** column. A check mark indicates that the Pset is enabled and attached to the object.
8. Select the custom Pset properties and assign values to the selected properties.
9. Click **OK** to assign the IFC data to the object.

   The IFC data can be reviewed or edited by clicking **IFC Data** from the Object Info palette of a selected IFC object or entity, or by selecting the object or entity and selecting **AEC > IFC Data** (Vectorworks Architect workspace) or **Landmark >Architectural > IFC Data** (Landmark workspace); see "Viewing and Editing IFC Data" on page 1748.

The custom property set is saved in the current file. To use a custom property set in another file or project, export the record format to another file and repeat the manage custom property sets process.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

IFC Format Interoperability
Assigning IFC Data to Objects
Record Formats
Creating IFC Schedules
Importing IFC Files
Exporting IFC Projects

A L Creating IFC Schedules

Before you export an IFC project, check that objects have appropriate IFC data associated with them. In the Vectorworks Architect product, the following schedules with IFC data can be added to the drawing with the **VA Create Schedule** command, or you can create your own schedules (see "Records and Schedules" on page 1859).

- **Objects with IFC Entity** gives a count of IFC objects for each entity type on each layer. The object class, name, and type also display, if available.
- **Objects with IFC Entity - Specific** lists each IFC object, broken down by entity type (walls, slabs, roofs, doors, windows, stairs, and columns). The object layer, class, name, and description also display, if available.
- **Objects without IFC Entity** lists each non-IFC object. The object layer, class, type, and type name also display, if available.

Alternatively, from the Resource Browser, open the default architectural reports file from the [Vectorworks]\Libraries folder that is included with the Vectorworks Architect product (see "Resource Libraries" on page 219). Drag the appropriate IFC worksheet to the drawing.

To edit the worksheets after they have been created, see "Using Worksheets" on page 1319.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

IFC Format Interoperability

A L Importing IFC Files

An IFC project, including one which contains multiple buildings or large information sets, can be imported into a Vectorworks file. The units of the imported file are determined by the Vectorworks file. The presentation layers or CAD layers from an imported IFC file are assigned to corresponding Vectorworks classes.

Stories and elements in the IFC file can be filtered during the import, to remove irrelevant objects. Filtering an import can significantly lower the file size, making both the import process and subsequent collaboration more efficient.

To import an IFC file:

1. Select **File > Import > Import IFC**.

Alternatively, click the file to import and drag it into a window where a Vectorworks document is open.

2. Select the .ifc, .ifczip, or .ifcxml file to open, and click **Open**.

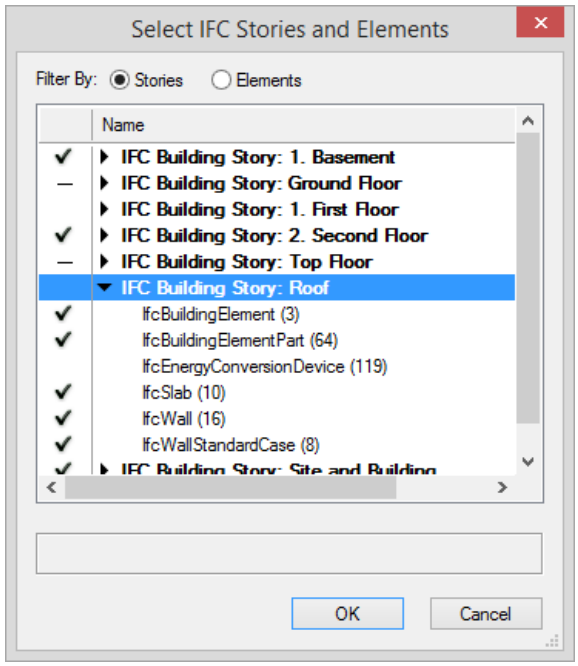
   The Select IFC Stories and Elements dialog box opens.

3. All stories and elements are imported by default. If you want to apply a filter to the import, choose whether to filter by **Stories** or **Elements**.

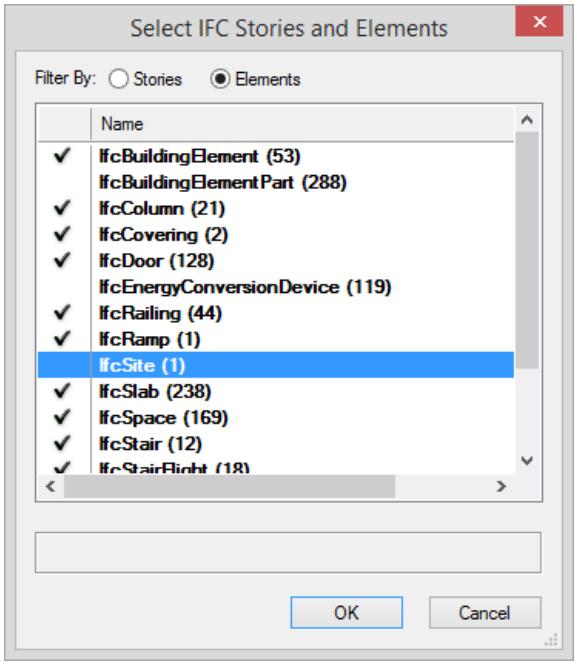
4. If **Stories** is selected, a list of all stories in the document displays; a check mark to the left of each story indicates that it is to be imported. To omit one or more whole stories and any associated elements from the import, select the story and click to remove the check mark.

   Alternatively, to omit only specific elements on specific stories from the import, click the disclosure arrow to the left of a story name to display the list of IFC elements associated with that story. Select the element(s) to be omitted and click to remove the check mark. The deselected elements are omitted only from that story.

   On stories with a combination of selected and deselected elements for import, the check mark is replaced by a dash.



If **Elements** is selected, a list of all IFC object types in the file displays; the number of elements of each type displays in parentheses to the right of the object type. A check mark to the left of each item in the list indicates that all are to be imported by default. Select the element(s) to be omitted from the import, and click to remove the check mark. The deselected elements are not imported on any story.



If you deselect some items from the elements list, and return to the stories list, stories that contain deselected elements display a dash rather than a check mark, indicating that some elements are not to be imported.

5. Click **OK**.

   The IFC file is imported.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

IFC Format Interoperability
Assigning IFC Data to Objects
Exporting IFC Projects

Exporting IFC Projects

Vectorworks project files can be exported to .ifc, .ifczip, and .ifcxml formats.

Export to IFC Versions 2x2 or 2x3 is supported. The export of a project to an IFC file is based on specifying the geometry and associated data needed for the use of the exported file. This specification is called a Model View Definition (MVD), which is a subset of all the geometry and data in a building model. MVDs have been created by international, national, or local groups to standardize information exchange for various purposes, such as design coordination, collision/clash detection, structural analysis, and element quantity analysis. You can use MVDs to automate the export, or manually define a custom set of geometry and data to be exported.

To export a Vectorworks file to IFC:

1. Select **File > Export > Export IFC Project**.

   The Export IFC Project dialog box opens.

2. Specify the overall project export options, site information, and author data for each pane of the Data tab. Many of the fields are required by the IFC specification; they are automatically mapped to their IFC equivalent for inclusion in the IFC file header information.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Export Options	Sets export parameters for the file
File Format	Select whether to export an .ifc, .ifczip, or .ifcxml file
IFC Version	Select the IFC version
Model View	Select the MVD that specifies the geometry and data required for export. • CV 2.0 - Architecture is an official buildingSMART International MVD for exporting architectural models for coordination with structural and building services models. • Extended Vectorworks Model View allows you to customize the geometry and data to be exported in conjunction with the **Export Options** for data exchange workflows without a standard or official MVD. • IES VE is a custom MVD to exchange model information with the IES-Virtual Environment platform to conduct building energy performance simulation and analysis. • COBie/Facilities Management MVD enables the handover of graphic and non-graphic facility management information to the building manager. It includes the international version of COBie/COBie2 exchange requirements. • Simplified Geometry exports walls as IfcWallStandardCase, merging all components into one extrude. Slabs are exported as multiple slab instances (one slab for each component). The model view and any add-on views are included as information in the IFC file header.
Add-on Views	Select optional information to be added to the model view at export. • Quantity Take Off writes element quantity data to specific IFC data fields for use by costing/estimating applications. • 1st Level Space Boundaries exports planes that represent the contact surface between a Space object and an adjacent surface, like a wall, floor, or ceiling. These boundary planes will also indicate the placement and extents of openings, such as doors and windows.
Export Visible Objects Only	Exports objects within visible classes only (the selection of layers for export is handled by the Layer Mapping options). When deselected, exports any objects required by the **Model View** selection.
Export Site Model	For drawings with a site model, this option exports the site model geometry and enables site model layers to be mapped on the Layer Mapping tab
Export Custom Property Sets	Exports custom Psets that were specified in the Manage Custom Property Sets dialog box for objects' IFC data, for models not explicitly defined in the Model View definition, or when using the extended Vectorworks model view
Export Solids Operations as BREPs	Depending on the receiving application and data exchange requirements, solids operations may need to be simplified to a surface model, also known as a boundary representation (BREP) to display correctly and be properly identified

Parameter	Description
Export Text in Unicode Format	The Unicode format is more reliable for double-byte characters (text representing characters from such alphabets as Japanese, Chinese, and Cyrillic, for example) but it must also be supported by the receiving application
Export Walls/Slabs by components	Exports the components of walls and slabs as sub-elements, for building material estimation purposes
Project Data	
Name	Type the project name
Phase	Enter the construction phase information
Project Type	Specify the type of project
Building Permit ID	Enter the building permit information
Gross Area (Planned)	Enter the total planned area for the entire project
Author Data	Enter the information for the primary author of the file and select the author's professional role. A custom role can be specified; select USERDEFINED and enter the role title.
Author Addresses	Enter pertinent information related to the physical location of the author's place of business
Organization Data	Enter pertinent information related to the professional organization in charge of the project
Organization Addresses	Enter pertinent information related to the physical location of the organization's place of business. If the author address data and the organization address data are identical, click **Copy Author Addresses** to complete the telecom and postal information automatically. Subsequent edits made to either pane remain unique to the pane.
Site Data	Specify site information for the project location, regardless of whether site model data is selected for export
Site Address	Enter address information for the project site
Building Data	Specifies the IFC properties associated with the building
ID/Name/Description	Enter the building's ID, name, and description
Gross Planned Area	Specifies the total planned area for the building, used for programming purposes
Reference Elevation/Site Reference Elevation	Enter the elevation above sea level for the building and the building site
Occupancy Type	Specifies the occupancy classification, defined according to the presiding national building code
Primary Construction Type	Specifies the main fire use for the building, assigned from the fire use classification table as given by the relevant national building code
Secondary Construction Type	Enter any ancillary fire use for the building, assigned from the fire use classification table as given by the relevant national building code
Sprinkler Protection	Indicates whether the building has sprinkler protection, and if so, whether it is automatic (the selection is reflected on the Layer Mapping tab, where it can be overridden for an individual story)
Year of Construction	Enter the year when the building was built

Parameter	Description
Landmark Status	Indicates a building with special historical landmark designation
Building Address	Enter address information for the building; if the site address and building address are identical, click **Copy Site Address** to complete the postal information automatically. Subsequent edits made to either pane remain unique to the pane.

3. Click the Layer Mapping tab to specify which layers to export, and the **Story Name** to assign to them at export. If specified, this includes the site model. For a project initially set up with stories, mapping is done automatically so that the layers assigned to the building stories are automatically included in the **Mapped Layers** list and are mapped to an appropriately named story (Vectorworks Architect required). Mapping can also be controlled manually.

   Specify the associated building or site data for each of the mapped layers.

Click to show/hide the parameters.

Parameter	Description
Vectorworks Story Name	Specify the mapping options for each story, and for the site model, if it was selected for export on the Data tab. As each story is selected, its associated layers display for mapping. Files set up with a story structure automatically map the story layers for export.
IFC Story Name	Displays the name assigned to the currently mapped layers for export; change the story name if necessary
Available Layers	Lists Vectorworks layers available for mapping; lists the layers associated with the number of stories and the story selected in the Story/Site Selection list
Number of Stories	Displays the number of stories in the file, which determines the number of stories available in the Story/Site Selection list. If the number of stories entered here is different from the number of stories set up for the file, additional layers display for possible mapping for the current story. This allows greater export flexibility, to override the story structure in the file and assign certain layers to different stories at export.
Mapped Layers	Lists layers that have been selected for export; they are associated with the **Story Name** entry
Add >	Adds available Vectorworks layers to the **Mapped Layers** list for export
< Remove	Removes mapped layers from the export list, returning them to the **Available Layers** list
Mapped layer data	Specifies IFC properties for selected mapped story layers
Above Grade	Indicates whether the story is fully above ground
Entrance Level	Indicates whether the story includes an entrance for the building
Sprinkler Protection	Specifies whether the story includes sprinkler fire protection, and if so, whether the sprinklers are automatic; initial settings match the sprinkler protection setting on the Data tab
Planned Areas, Gross/Net	Indicates optional gross and net planned areas, used for design and programming purposes

4. Click **OK** to export the project.

   Indicate the file name and location in the Save As dialog box.

IFC Format Interoperability
Assigning IFC Data to Objects

Importing IFC Files
Setting Up the Building Structure with Stories

Viewing BCF Files

BIM Collaboration Format (BCF) allows collaborators to exchange comments between software applications without exchanging the entire BIM model. Issues, proposals, and change requests about a project are “topics” in the BCF file, linked to specific objects in the BIM model. The BCF file only contains comments; it does not modify the model.

To view a BCF file:

1. Open the Vectorworks file that is associated with the BCF file. For best results, the BIM model in the Vectorworks file should exactly match the version of the model that was used when the BCF file was created.
2. Select the **Open BCF Viewer** command from the appropriate menu:
   - Architect workspace: **AEC > Open BCF Viewer**
   - Landmark workspace: **Landmark > Architectural > Open BCF Viewer**
3. If no BCF file is currently imported into this Vectorworks file, the Import BCF dialog box opens. Select the desired .bcfzip file and click **Open**. If the BIM model in the current Vectorworks file is different from the model the BCF file was created with, an alert displays.
4. The BCF Viewer dialog box opens and displays data from the imported file.

Click to show/hide the parameters.

Parameter	Description
List of Topics	All topics in the file display, along with the date each was added; click a topic to view the data associated with it in the bottom of the dialog box
Topic Snapshot	Displays an image of the IFC objects described in the selected topic; since the snapshot was created in a different software application, it will not match the view in the Vectorworks file exactly
Comments on Selected Topic	All comments associated with the selected topic display; for each comment the following information displays: • Date of comment • Abbreviated Comment text • Status • Author • Additional Status Text
Comment	Displays the full text of the comment

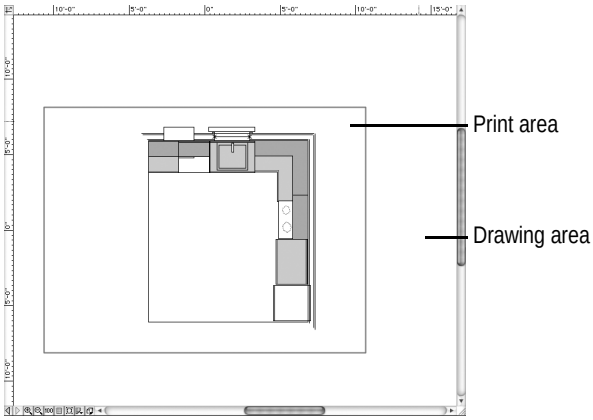
5. Click **OK** to switch the view in the Vectorworks application to display the objects described in the selected topic. The related objects are selected, and the view is similar to the snapshot.
6. To open another BCF file related to this project, click **Import BCF,** and select the BCF file from the Import BCF dialog box. The previous BCF file closes, and the new one opens.
7. When you open a BCF file, saved views, a symbol folder, and worksheets are added to the Vectorworks file. When you are done reviewing the BCF file, click **Close BCF** to remove the BCF data.

Printing and Publishing

Printing

The Print Area

Within the drawing area is a colored border which defines the print area. Anything within the print area is printed and anything outside of the print area is not printed.



The settings from both the Page Setup and Printer Setup dialog boxes determine the number of sheets of paper (pages) required to print the entire drawing on the selected printer.

This allows the same drawing to be printed or plotted to numerous sources without readjusting the drawing. For example, if you enter the dimensions of a “D” size sheet of technical paper in the Printable Area of the Page Setup dialog box, and select “Letter” paper size in the Printer Setup dialog box, the program determines that the drawing needs to be “tiled” across 12 sheets of 8-1/2” x 11” paper to a standard laser printer. The number of 8-1/2” x 11” pages required for printing horizontally and vertically are automatically updated in the Page Setup dialog box. To send this same drawing to a plotter, change the settings in the Page Setup dialog box for a plotter, and the program converts the pages required accordingly.

The color of the print area border can be changed; see “Configuring Interactive Display” on page 116.

> The page setup (including the printable area) is specified individually for each sheet layer (see “Setting Sheet Layer Properties” on page 170).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Page Setup
Printer Setup
Printing a File
Setting Quick Preferences

Page Setup

The Page Setup dialog box settings determine the number of sheets of paper (pages) required to print the entire drawing on the selected printer. It is also used to specify whether to display the gray drawing boundary box representing the print area for design layers, and whether to display page breaks in the drawing file.

To change the page setup settings:

1. Select **File > Page Setup**.

   The Page Setup dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Pages	
Horizontal	Specify the number of pages in the horizontal direction
Vertical	Specify the number of pages in the vertical direction
Show page boundary	Select to display a gray border around the perimeter of the print area for design layers (this setting is selected by default); alternatively, press Alt+B (Windows) or Command+B (Mac) to toggle selection in this field. This setting also affects the visibility of the page boundary in a unified view. Sheet layers always display the page boundary.
Show page breaks	Indicates how the drawing is divided over each printable page; for design layers, the page boundary must be displayed in order to see the page breaks (this setting is selected by default)
Show watermarks	If the file is created in an educational version of Vectorworks software, it includes a watermark. This option is selected by default and indicates whether to display watermarks on screen, at the top and bottom of every page boundary; the page boundary must be displayed in order to see the watermarks. The title bar clearly indicates watermarked files. A watermarked file always prints the watermark, regardless of the display option set here.
Printable Area	
Choose size unavailable in printer setup	If the desired paper size is not available in the Printer Setup dialog box (see "Printer Setup" on page 1762), select this option, and then select the desired paper **Size** from the list; this is useful when sending files to a print bureau that has different paper sizes available than your printer
Size/Width/Height	To manually specify the paper size, select the appropriate drawing units (inch or millimeter) and enter the paper **Width** and **Height**
Printer Setup	Accesses the Printer Setup dialog box for specifying the printer, paper size, drawing scale, and orientation; see "Printer Setup" on page 1762

2. Specify the page setup options, and click **OK**.

Printer Setup

The Printer Setup dialog box settings determine the printer, paper size, scale, and orientation of the drawing.

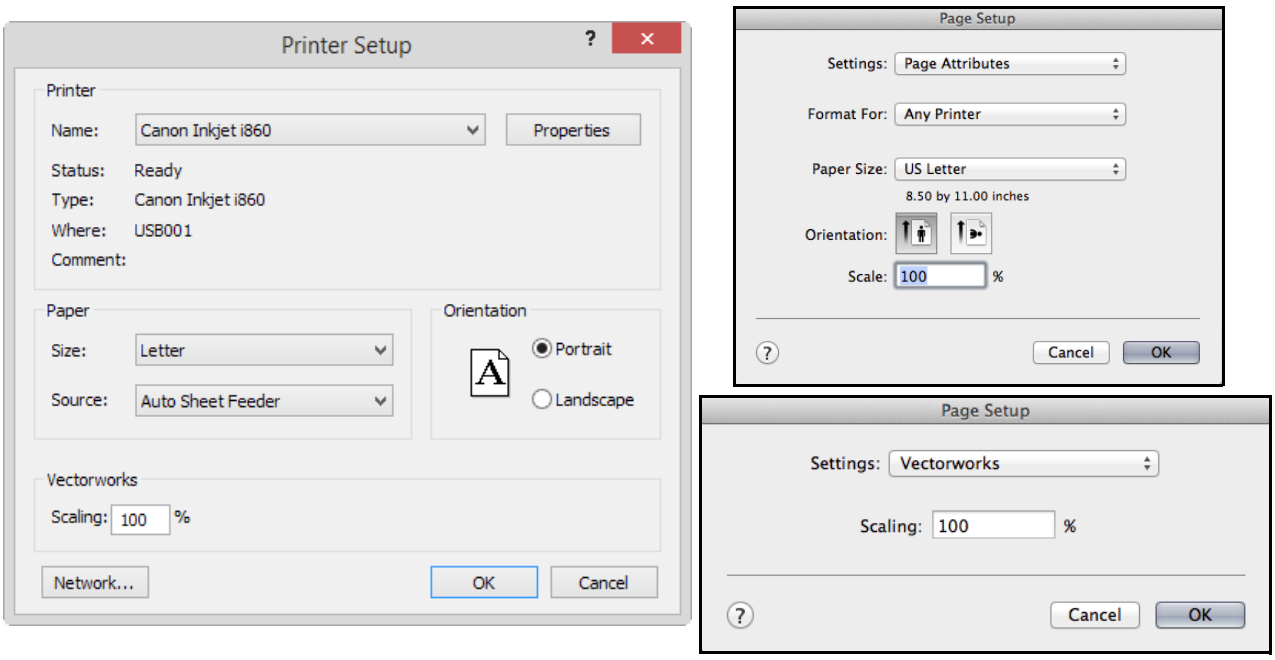
To change the printer setup settings:

1. Select **File > Page Setup**.

   The Page Setup dialog box opens.

2. Click **Printer Setup**.

   In the dialog box that opens, configure the selected printer options.



3. Scaling resizes the entire drawing by a specified percentage. To access the **Scaling** parameter in the Mac Page Setup dialog box, select Vectorworks from the **Settings** list.

Selecting a printer in Printer Setup (Mac) only changes the drawing's settings; to change the printer used for the drawing, access the Mac System Preferences.

Printing a File

The program can print or plot to any device that is selected in the Printers and Scanners System Preferences (Mac) or Devices and Printers Control Panel item (Windows). The actual parameters of the printed or plotted file are determined by the printer setup settings.

All visible objects, classes, and layers within the print area of the Vectorworks file are printed.

If Vectorworks Design Series is installed, you can also use the **Publish** command to print a batch of files. See "Batch Publishing" on page 1765.

To print a file:

1. Select **File > Print**.

   The Print dialog box opens; the options that display depend on which printer or plotter is selected.

   To display additional print options on the Mac, click **Show Details.**

2. Specify the print options, such as which pages to print and how many copies to make. The following Vectorworks print settings are available:

Platform	Setting	Description
Mac	Application resolution (DPI)	Sets a resolution for the application output; this affects only Vectorworks geometry and any PICT or PDF objects that must be rasterized. The resolution for bitmaps, rendered design layers, and viewports are controlled elsewhere; see “Setting the Print Resolution” on page 1765 for details. Either select a recommended DPI value for this printer type (from the drop-down list), or manually enter a value. On Mac systems, this always defaults to a minimum of 300 DPI, for optimum printing results.
Windows and Mac	Gray level for grayed layers and classes	Adjusts the level of gray when printing with grayed layers and or/classes
Windows	Enable special processing for transparent color bitmaps	When GDI+ imaging is disabled in the Vectorworks display preferences on Windows, some printers (including PostScript printers) cannot support transparent color for raster images, which can affect bitmaps and picture objects that have the fill style set to **None**. Select this option to use a more detailed (and slower) imaging process to print transparency properly.
Windows and Mac	Print current view only (Mac) or Current View (Windows)	Prints the view currently displayed in the file window; this view is scaled larger or smaller to fit the selected page size
Windows and Mac	Print patterns at on-screen resolution	Prints fill patterns at approximately the same size as they appear on the screen at a 100% zoom. Deselect this option to use the printer resolution, which is usually much higher, making the patterns print much smaller. This prints patterns much faster on Windows, if GDI+ imaging is disabled and you have a non-PostScript printer.
Windows	Rasterize print output	Prints the drawing as a raster bitmap. Select this option when the printer memory is insufficient, or when image problems occur (for example, to print design layer transfer modes on PostScript printers).
Windows and Mac	Update visible out of date viewports prior to printing	Automatically updates any visible, out-of-date viewports before printing
Windows and Mac	Reset all plug-in objects that require a reset prior to printing	Automatically resets plug-in objects that require an update (such as data stamps) before printing
Windows and Mac	Recalculate worksheets prior to printing	Automatically recalculates worksheets before printing

3. If the **Rasterize print output** option is selected, specify the edge of the image onscreen that is the first edge to come out of the printer.

The first edge to come out of the printer depends on the printer driver, the page orientation, and the rotation options that are supported by the driver. It cannot be predicted by the program. If the wrong page edge is selected, some portions of the drawing may fail to print if the printer runs out of memory.

The **Rasterize print output** method can be significantly slower than other print methods.

4. Click **Print** (Mac) or **OK** (Windows) to print.

Setting the Print Resolution

The resolution at which Vectorworks objects are printed is controlled by different settings, depending on the type of object and on whether imaging preferences are enabled.

- A raster rendered viewport is controlled by the **Raster Rendering DPI** setting of its sheet layer (which does not affect vector geometry or bitmaps that are inside the viewport). See “Setting Sheet Layer Properties” on page 170.
- A raster rendered design layer is controlled by the **Design Layer Raster Rendering DPI** on the Resolution tab in the Document Preferences. See “Resolution Preferences” on page 62.
- Vectorworks geometry is controlled by the resolution selected from the Print dialog box.
- Rasterized PDF objects are controlled by the resolution selected from the Print dialog box. On Windows, if **GDI+ imaging** is not enabled, PDF objects are rasterized.

Click here for a video tip on this topic (Internet connection required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Print Area

Batch Publishing

The **Publish** command exports a series of sheet layers and/or saved views from the current drawing and externally-referenced files to DXF/DWG or DWF format. Vectorworks Design Series users can also export to PDF or Image formats, or output to a printer with this command.

The publish list and options can be saved as a set. The list order can be changed, and each item can be published in either color or black and white. If the Issue Manager is in use (Vectorworks Architect or Vectorworks Landmark required), sheets and views can be sorted by issue date for quick publishing.

To batch publish sheet layers and/or saved views:

1. Set up the sheet layers and/or saved views as desired for publishing.

   Set the visibility of layers and classes in the viewports/views. All visible and grayed layers are published. Invisible layers and classes are not printed, or published to PDF or DWF; they can be published as invisible DXF/DWG layers, if the option is selected in the DXF/DWG Export Options dialog box. Each sheet layer or saved view is published as a separate file.

2. Select **File > Publish**.

   The Publish dialog box opens.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Sheets and Saved Views Available (Available list)  sheet layer  saved view	Lists the sheet layers and saved views present in the current file (indicated with <Active File> in the **File Name** column) and, if specified, within the selected folder (indicated with a file name and path). Click in a column header to sort the sheets and views by the column parameter. For example, click in the **Issue Date** column to sort by the Issue Manager issue date (Vectorworks Architect or Vectorworks Landmark required). Sheets and saved views from restored sets that cannot be found display in red and cannot be published.
Show All Items	As items are added to the publish list, keeps them in the available list. If you need to publish an item to multiple formats (for example, to both print (Vectorworks Design Series required) and export to DWF), select this option to show all sheet layers and saved views in the file, whether or not they have been selected to be published. Deselecting this option will cause an item to disappear from the Available list when you add it to the Publish list.
Folder path	Displays the most recently selected project folder and file path, for including sheet layers and saved views from other Vectorworks files within the folder
Select Folder	Opens the Choose Project Folder (Mac) or Select Folder (Windows) dialog box; select a project folder, including any subfolders, that contains sheet layers and saved views to be published. Once a project folder has been selected, all saved views and sheet layers from Vectorworks files within the folder (and any subfolders) are automatically added to the available list. Files must be the same version as the active file.
Add New Items as	Sets the default **Publish To** value for each item that is added to the Publish list (DWG/DXF or DWF). If Vectorworks Design Series is installed, Print, PDF, and Image options are also available.
Add 	Adds one or more selected item(s) from the Available list to the end of the Publish list
Add All 	Adds all sheet layers/saved views in the Available list to the end of the Publish list
Remove 	Removes one or more selected item(s) from the Publish list
Remove All 	Removes all sheet layers and saved views from the Publish list
Set to Publish	Select a previously saved set to publish
Save a Set	Saves the current publish set in the current file, to be restored later; see “Saving a Publish Set” on page 1768
Manage Sets	Manages saved file sets, and restores a previously-saved set of sheets and views for publishing; see “Restoring a Saved Publish Set” on page 1768

Parameter	Description
Sheets and Saved Views to Publish (Publish list)  color  black and white	Lists the sheet layers/saved views to be published; to change the list order, use the cursor to drag and drop the items in the # column. Click **Color** for each item to toggle between color and black and white.
Publish To	Select one or more items in the publish list and then select from this list to change the output format (DWG/DXF or DWF). If Vectorworks Design Series is installed, Print, PDF, and Image options are also available.
Options	Opens the appropriate dialog box to set the options for the currently selected item: • DWG/DXF and DWF: see “DXF/DWG and DWF Export Options” on page 1734 • Print (Vectorworks Design Series required): see “Printing a File” on page 1763 • PDF (Vectorworks Design Series required): see “Exporting PDF Files” on page 1690 • Image (Vectorworks Design Series required): see “Exporting an Image File” on page 1683; some image export options are not available when publishing All items of a specific type that are added to the publish list will default to the user specified options if left unchanged.
Export PDFs as Separate Files (Vectorworks Design Series required)	For items to be published to PDF, exports each selected sheet layer and saved view as a separate PDF file; deselect this option to create a single PDF file only
Open Destination Folder upon Completion	After publishing, opens the containing folder for published files
Create Folder for Each File Type	Creates a folder for each file type within the destination folder Certain publish formats, such as DWG/DXF, require subfolders for output organization, even when this option is deselected.

3. Select available items from the available list on the left side of the dialog box, and use the buttons in the middle of the window to move them to the publish list on the right side of the dialog box. The **Publish To** column in the right pane indicates the current output setting for each item. To publish an item to multiple formats, **Show All Items** must be selected; add the item to the publish list multiple times, and specify a different format for each one.
4. Verify that the output options for each item are set appropriately, and then click **Publish**.
5. If the Publish set has not been saved, you are prompted to save it.
6. If PDF, DWG/DXF, DWF, or Image is a selected publish format, the Select Folders dialog box opens. Specify a location for the output folders.
7. Click **OK**. The output files are printed or created based on the selected settings. Within the specified output folder, a subfolder is created for each type of output file. A publish log file is created in the same location, which summarizes the results of the publish job.

Click here for a video tip about this topic (Internet access required).

Saving a Publish Set

You can save the current publish settings to be reused later. The set of sheet layers and saved views, along with their color settings, list order, and publish method, are saved in the file. If the file set contains sheet layers and views from a selected folder, specify how to save the path relative to the current file.

The PDF, Image, and Print publish options are available in Vectorworks Design Series only. Items that were saved with PDF, Image, or Print options will not display if the saved set is opened with Vectorworks Fundamentals.

To save the export list as a set:

1. In the Publish dialog box, click **Save a Set**.

   The New Saved Set dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Name	Enter a unique name for the set
Absolute path	Sets an absolute path to files used by the saved set. Use the absolute path when the location of the referenced files with respect to the current file is not going to change.
Path relative to the active document	Sets a relative path to files used by the saved set. Use the relative path when the files might be moved to another computer or platform; as long as the relative path between the files remains the same, the referenced files can be found. All files must be saved on the same volume or server to select this option.
Open Destination Folder upon Completion	Select to open the containing folder for the saved set
Create Folder for Each File Type	Select to create a folder for each file type in destination folder
Export PDFs as Separate Files (Vectorworks Design Series required)	Select to publish each sheet layer or saved view as a separate PDF file

2. Click **OK**. The settings are saved with the file and are available for future publish jobs.

Restoring a Saved Publish Set

To restore a saved publish set:

1. In the Publish dialog box, click **Manage Sets**.

   The Manage Saved Sets dialog box opens.

Click to show/hide the parameters.

Parameter	Description
List of saved sets	Lists the currently saved sets by name; select a set to edit, recall, or delete it
Edit	Opens the Edit Saved Set dialog box; rename the set and/or change the file path style (relative or absolute)
Delete	Deletes the currently selected saved set(s)

Parameter	Description
Recall	Restores the list of saved views and sheet layers, along with their settings, to the export list in the Publish dialog box

2. Select a saved set.
3. Click **Recall**. The settings for the selected saved set display in the Publish dialog box.

   If the saved set references external files and a file, sheet layer, or saved view from the set cannot be located, you are prompted to manually locate a missing file, or warned that sheets or views are missing. Missing sheets and views display in red in the Publish dialog box, and are not included in the publish job.

Using Scripts

Scripting is one method of customizing the Vectorworks program to suit a wide range of design needs. Scripts can create anything from simple tools that assist with the most tedious drafting tasks, to sophisticated solutions for demanding designs. Three scripting options in the Vectorworks program include:

- The Software Developer's Kit (SDK) in the C++ language
- Python® programming language
- VectorScript™, a lightweight Pascal-like programming language

This section discusses scripting, but not the SDK. For detailed developer-oriented documentation related to the SDK, Python, and VectorScript, please navigate to http://developer.vectorworks.net

The Vectorworks program provides several methods for creating, managing, and executing scripts. While scripting languages provide a rich set of development tools when creating scripts from scratch, there are also several commands that allow you to create scripts without direct knowledge of programming. The most basic of these methods is to select the **File > Export > Export Script** command. The command creates a script from the file's contents and saves it as a Python or VectorScript file in the designated location. Select the **File > Import > Import Script** command to execute the script from a saved file in a new, automatically created design layer.

The **Custom Selection**, **Custom Tool/Attribute**, and **Custom Modification** commands provide the ability to create useful scripts directly.

In addition to these automated methods of creating scripts, the traditional method for creating and saving scripts (and a feature of Vectorworks since its original release as MiniCad), is to write document scripts, which are saved as script resources and are stored in script palettes contained within a file. Both document scripts and script palettes can be created, accessed, exported, imported from another file, renamed, and deleted through the Resource Browser.

Scripts can also be created and stored in plug-ins. Plug-ins are a component of a workspace, and can be accessed from any file. Scripts in plug-ins can be used as menu items, tools, or parametric objects. Plug-ins are created and maintained using the Plug-in Manager, which is accessed by selecting **Tools > Plug-ins > Plug-in Manager**.

Finally, scripts can be purchased from third-party vendors and installed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Custom Selection Scripts

Custom selection scripts allow objects in the current file to be selected or deselected according to user-specified criteria. The wide array of options in the command allows for very specific filtering and selection according to the defined selection requirements. No scripting knowledge is necessary.

The layer options must be set to Show/Snap/Modify Others to select objects on another layer.

To create a custom selection script:

1. Select **Tools > Custom Selection**.

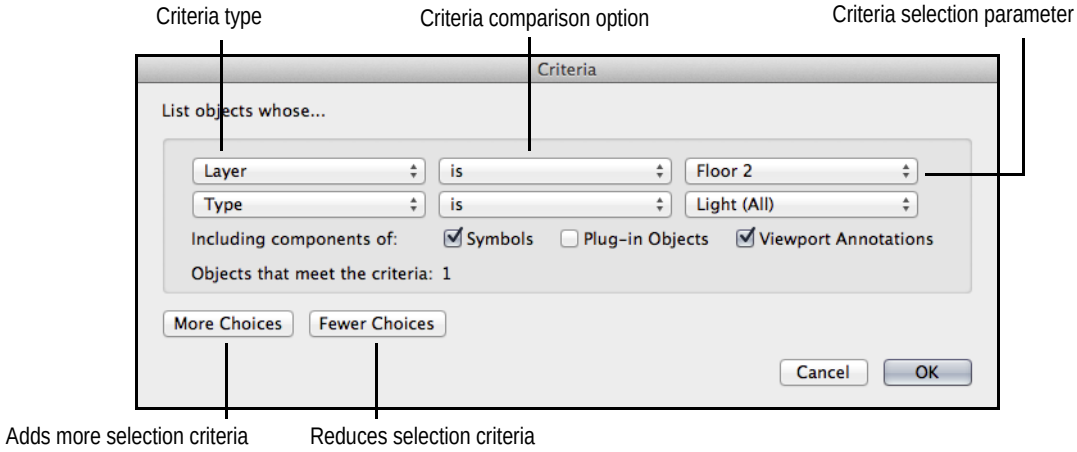
   The Custom Selection dialog box opens. Choose the desired options for the selection operation.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Select	Selects the objects matching the specified criteria without affecting the existing selection status of other objects
Select Only	Deselects all objects prior to performing the selection operation
Deselect	Deselects any objects matching the specified criteria without affecting the selection status of other objects in the file
Execute Immediately	Performs the specified selection operation immediately upon exiting the Criteria dialog box; the operation criteria are not saved for future use
Create VectorScript	Saves the criteria and command options as a VectorScript that can be executed later
Create Python Script	Saves the criteria and command options as a Python script that can be executed later

2. Click **Criteria** to specify the selection criteria for the operation.

   The Criteria dialog box opens.



3. Choose the desired criteria options. The number of objects in the file that meet the criteria displays.

   Specify additional options by clicking **More Choices**. To remove the most recently added option, click **Fewer Choices**.

   Specify whether to also include objects that are part of symbols, plug-in objects, or viewport annotations.

4. Click **OK**.

   If the **Execute Immediately** option was chosen, the selection operation is now performed. If one of the create script options was chosen, when prompted, name the script palette (if none exists) or select an existing script palette, and name the selection operation script. The script is saved in the script palette, and can be run as described in “Running Scripts” on page 1780.

   Click here for a video tip about this topic (Internet access required).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Entering Formulas in Worksheet Cells
Using Scripts
Opening a Script Palette
Running Scripts
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Custom Tool/Attribute Scripts

Custom tool/attribute scripts record the current attribute and file settings in a script format for later use. These custom settings can be combined with the active tool to allow a specific set of attributes, such as fill style or pen color, to be reactivated for use with the tool. For example, a dashed line style and red pen foreground color can be saved with the line tool in a script for drawing red dashed lines. No scripting knowledge is necessary.

To create a custom tool/attribute script:

1. Select **Tools > Custom Tool/Attribute**.

   The Custom Tools dialog box opens.

2. Select which attributes should be saved in the script.

   The saved attributes are used when the script is executed and can be modified or reset.

Click to show/hide the parameters.

Parameter	Description
All	• **Checked box:** All attributes in the group are selected; click to deselect all attributes in the group to be saved to the script. • **Empty box:** No attributes in the group are selected; click to select all attributes in the group to be saved to the script. • **Box displays (Windows) or (Mac):** One or more attributes in the group are not selected; click to select all attributes in the group to be saved to the script.
Fill Attributes	Saves fill foreground and background colors and fill style
Pen Attributes	Saves pen foreground and background colors and pen style
Line Attributes	Saves line weight (thickness) and the presence and size of markers
Text Attributes	Saves text attributes including font, size, style, spacing, and justification (alignment)
Other Attributes	Saves other attributes such as classes, layers, symbols, tools, and SmartCursor settings (snapping parameter settings, as well as which snapping tools are active on the Snapping palette)

3. Click **OK**.
4. When prompted, name the script palette (if none exists) or select an existing script palette, and name the attribute settings script.

   The script displays in a script palette, and can be run as described in “Running Scripts” on page 1780.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Scripts
Opening a Script Palette
Running Scripts

D Creating Custom Modification Scripts

Custom modification scripts can select multiple objects and modify their common properties and attributes in a single operation. Objects can be selected regardless of layer, layer scale, class, or current selection or visibility status. Several levels of selection criteria make it easy to target specific object sets for modification. No scripting knowledge is necessary.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To create a custom modification script:

1. Select **Tools > Custom Modification**.

   The Custom Modification dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Execute Immediately	Performs the specified selection operation immediately upon exiting the Custom Modification dialog box; the operation criteria is not saved for future use
Create VectorScript	Saves the criteria and command options as a VectorScript that can be reused as needed
Create Python Script	Saves the criteria and command options as a Python script that can be reused as needed
Selection Criteria	Determines the objects to select and modify based on the selected criteria
Including components of	Applies the selection to objects that are grouped and/or included in viewport annotations
Objects that meet the criteria	Displays the number of drawing objects that will be affected
More Choices/Fewer Choices	Specify additional criteria by clicking **More Choices**. To remove the most recently added level of criteria, click **Fewer Choices**.

2. Select the desired criteria options. The number of objects that meet the criteria displays.

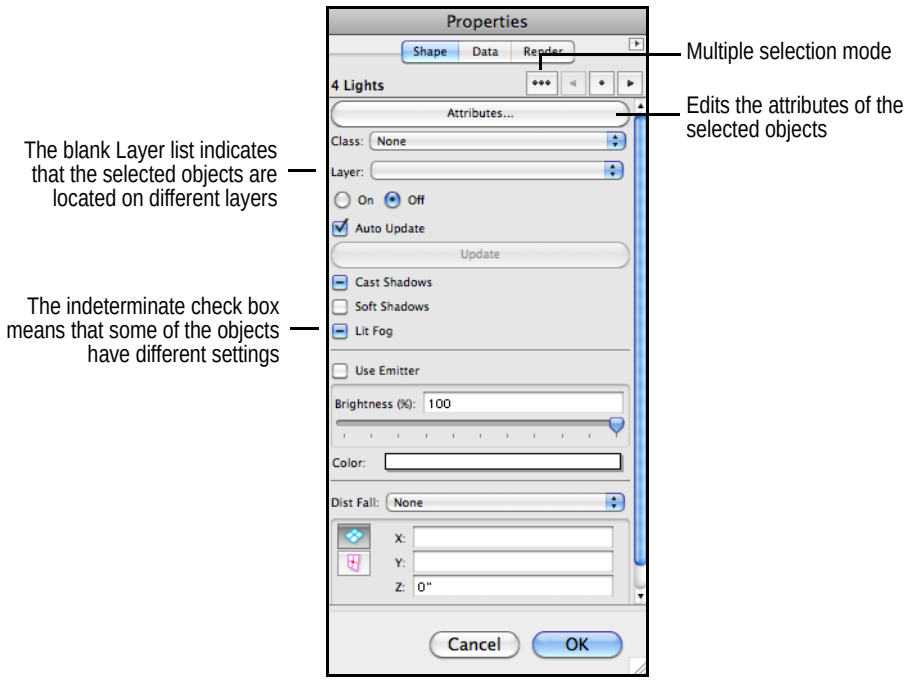
   When symbols are selected as the criteria and symbol folders are present in the file, click the button available near the symbol type to open the Select Symbol dialog box. Select the specific symbol, and then click **OK** to return to the Custom Modification dialog box.

3. Click **OK**.

   If the **Create Script** option was chosen, when prompted, name the script containing the modification commands. The script then displays in a script palette, and can be double-clicked to perform the operation. See “Creating and Editing Script Palettes and Scripts” on page 1775.

4. If the **Execute Immediately** option was chosen, the Properties dialog box opens.

   The common properties of the selected objects are displayed and can be modified. Similar to a multiple selection, if the parameter settings for the objects are different, they display as a blank list or as a check box of indeterminate state. Parameter edits apply to all selected objects when in multiple edit mode.



5. To edit the object attributes, click **Attributes**.

   The Attributes dialog box opens. Edit the available attributes of the selected objects, and click **OK**. See "The Attributes Palette" on page 1093 for more information.

6. Click **OK** to return to the drawing.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using Scripts
Opening a Script Palette
Running Scripts

Creating and Editing Script Palettes and Scripts

Document scripts can be created, saved, and executed within a Vectorworks file. Use the Resource Browser to manage script palettes as well as individual document scripts.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Script Palette
Opening a Script Palette
Creating Scripts
Editing Scripts
Using Scripts
Running Scripts

Creating a Script Palette

Scripts are organized and accessed from script palettes located in the Resource Browser. If no script palettes exist when a script is created, a new script palette resource is automatically created. However, creating a script palette ahead of time allows you to select an existing script palette when creating new scripts.

To create a script palette:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.

2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Script Palette**.

   The Assign Name dialog box opens. Enter the name of the script palette and click **OK**.

Opening a Script Palette

To open a script palette:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.
2. Select the script palette to be opened, and then select **Resources > Open**.

   Alternatively, open the palette from the **Window > Script Palettes** menu.

   The script palette opens. While open, scripts can be easily executed from the palette.

   Script palettes can be edited, renamed, deleted, exported, and imported from the Resource Browser.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Scripts
Editing Scripts
Using Scripts
Running Scripts
Using the Resource Browser

Creating Scripts

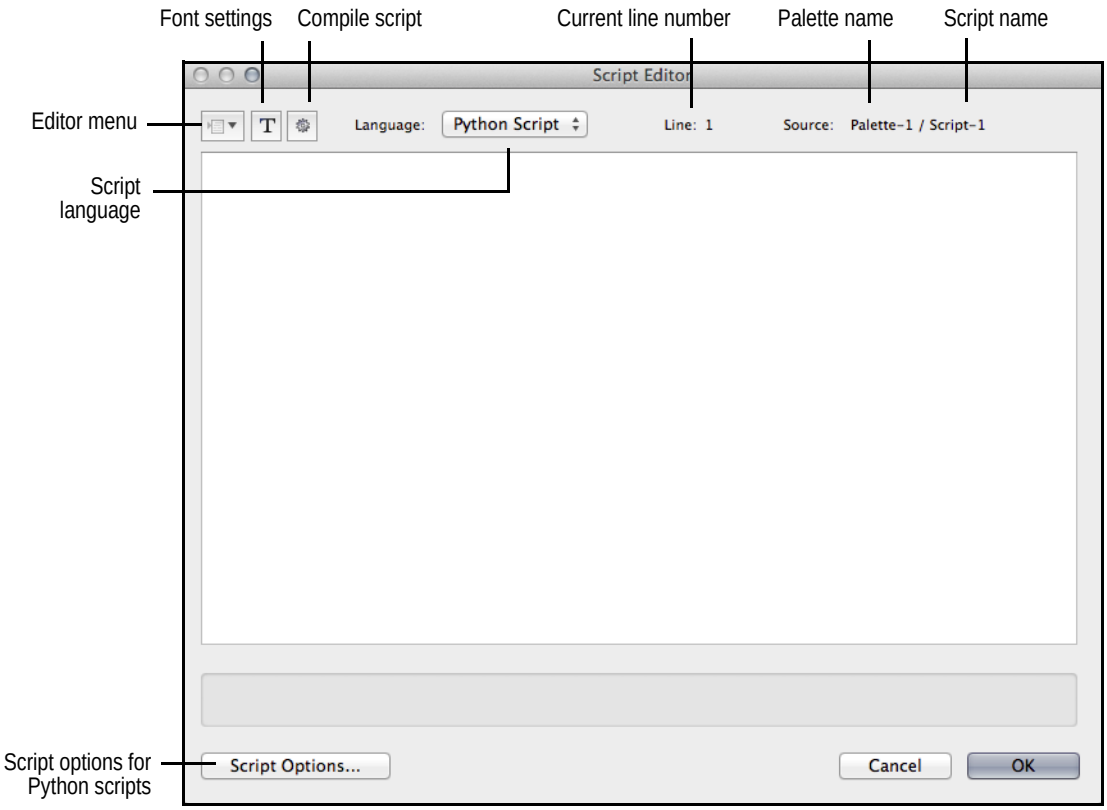
To create a script:

1. Select **Window > Palettes > Resource Browser**.

   The Resource Browser opens.
2. Select **Resources > New Resource** to display the list of new resource types.
3. Select **Script**.

   The Assign Name dialog box opens. Enter the name of the script and click **OK**.

   Newly created scripts are located by default in the active script palette (the palette which is open and active, or which is open in the Resource Browser). If multiple script palettes exist, but are not currently open, the Select Script Palette dialog box opens to select a location for the script. Select the palette to add the script to and click **Add**. If no script palette exists in the file, a new script palette must be created to contain the script.
4. The Script Editor opens to begin a script editing session. The editor provides a basic authoring environment for script development and maintenance. Its features allow you to create, edit, and compile scripts, browse available functions, view errors, and perform other tasks associated with creating scripts. Select the script language for the script: Python Script or VectorScript. Scripts are limited to 32,001 characters. For detailed developer-oriented documentation related to Python and VectorScript, please navigate to http://developer.vectorworks.net
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click to show/hide the parameters.

Parameter	Description
Editor menu 	
Procedure	Opens the Procedures dialog box, which lists all the available script functions/procedures by category and provides a prototype as well as a brief description of what operation is performed by the function/procedure. The list serves as both a reference and a means of building the code by pasting the relevant function/procedure into the editor. Select the procedure or function and click **OK** to add it to the current line of the script editor.
Criteria	Opens the Criteria dialog box, for automatically defining selection criteria in a script (see “Creating Custom Selection Scripts” on page 1771)
Tool/Attribute	Opens the Custom Tools dialog box, for saving the current tool and attribute state information into a script (see “Creating Custom Tool/Attribute Scripts” on page 1773)
Parameter	The **Parameter** command provides access to a plug-in objects’ parameter list for editing, when working on the script of a plug-in (see “Plug-in Definition” on page 1785)
Text File	The **Text File** command allows script source code to be imported from external text files
Font Settings 	Opens the Format Text dialog box, where the font settings of the text in the Script Editor window can be changed. Changes to the text formatting apply to all of the text in the Editor (formatting a selection of text is not possible). See “Formatting Text” on page 389 for more information on text formats.

Parameter	Description
Compile Script 	The **Compile Script** button allows a script to be compiled directly from the Script Editor without the need to execute the script. If errors exist within the script that prevent successful compilation, they will be displayed and can be resolved without the need to exit the script editor. While compiling VectorScripts reports all errors in the script code, compiling Python scripts reports only the syntax errors. For example, an undefined variable is not reported as an error for Python scripts, because this type of error is determined only during script execution. As a result, Python scripts report most errors when executed.
Language	Select Python or VectorScript for the script language
Script Options (Python scripts only)	Opens the Script Options dialog box, which lists folders (environment paths) where imported modules are located, for Python scripts that require external libraries. This dialog box can also be accessed from **Tools > Plug-in > Script Options**. • **Add**: Select a folder location to add it to the list • **Remove**: Deletes the currently selected folder location • **Open Folder in OS**: Displays the contents of the selected folder in either Windows Explorer or Mac Finder

5. When the script is complete, click **OK** to save it.

   Scripts can be edited, renamed, deleted, exported, imported, moved, and executed from the Resource Browser.

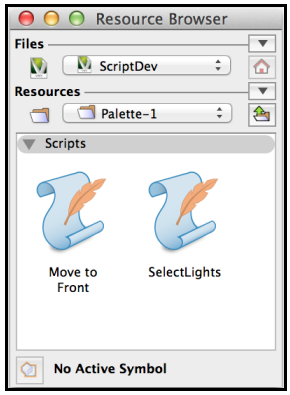
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Scripts
Running Scripts
Using the Resource Browser
Creating a Script Palette
Using Scripts

Editing Scripts

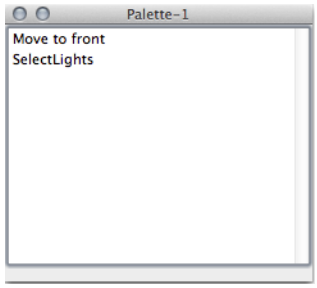
To edit an existing script from the Resource Browser:

1. From the Resource Browser, select the script to be edited. Scripts are nested in script palette folders.



2. Select **Resources > Edit**.

   Alternatively, open the script palette from the Resource Browser. Press Option (Mac) or Alt (Windows) and double-click on the script to be edited.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



3. The Script Editor opens, displaying the script source code. See “Creating Scripts” on page 1776 for a description of the Script Editor.

Moving a Script to Another Palette

To move a script from one palette to another:

1. From the Resource Browser, select the script palette containing the desired script, and then select **Resources > Open**. Alternatively, open the palette from the **Window > Script Palettes** menu.

   The source script palette opens.

2. Select the script palette where the script will be moved, and then select **Resources > Open**.

   The destination script palette opens.

3. Click on the script to move, and then press Command+X (Mac) or Ctrl+X (Windows). The script is removed from the palette and placed into the clipboard.
4. Click on the destination script palette, and then press Command+V (Mac) or Ctrl+V (Windows). The script is placed into the selected script palette.

Alternatively, move the script by selecting it from the Resource Browser and selecting **Resources > Move**. Specify the new script palette location for the script. See “Using the Resource Browser” on page 221 for more information.

Encrypting Scripts

A script can be encrypted and locked to prevent editing.

To encrypt a script file:

1. Select **Tools > Plug-ins > Encrypt Script**.

   The Choose Script File dialog box opens.

2. Locate and select the script file to encrypt.
3. Click **Open**.

   The Save Encrypted VectorScript Code Document dialog box opens.

4. Enter a new name for the script, and then select the location for saving the file.
5. Click **Save**.

   The script is saved in an encrypted format.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Running Scripts
Creating Scripts
Using the Resource Browser
Creating a Script Palette
Using Scripts
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Running Scripts

Document scripts can be executed in a variety of ways.

Running Scripts from the Resource Browser

To run a script from the Resource Browser:

1. From the Resource Browser, select the script palette containing the script to run.
2. Double-click on the script palette to open it.

   Alternatively, right-click (Windows) or Ctrl-click (Mac), and select **Enter** from the context menu to open the palette.
3. Select the script to run. Select **Resources > Run**, or right-click (Windows) or Ctrl-click (Mac), and select **Run** from the context menu.

   Alternatively, double-click the script resource to run it or drag the script resource into the drawing area.

Running Scripts from the Script Palette

To run a script from a script palette:

1. Select **Window > Script Palettes** and select the script palette containing the desired script.

   To open the script palette from the Resource Browser, select the script palette containing the desired script, and double-click to open it, or right-click (Windows) or Ctrl-click (Mac) on the palette, and select **Open** from the context menu.
2. From the script palette, double-click on the script to run it.

Running Scripts from Files

Some scripts may exist as text files only, if they were created in a separate text editor program or exported from the Vectorworks program. These scripts are not available from the Resource Browser.

1. Select **Tools > Plug-ins > Run Script** (or **File > Import > Import Script**).

   The Choose Script File (or Import Script Drawing Document) dialog box opens.
2. Select the desired script file and click **Open** to run the script.

Script Errors

If an error occurs during script execution, an alert dialog box opens.

To handle script errors:

1. From the alert dialog box, click **View Error Output**.
2. The Script Errors dialog box opens, displaying the nature of the error and the line(s) where the error occurred.
3. To edit the script, click **Edit Script**, or double-click on a script line to edit that line. The Script Editor opens for making corrections to the script.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Scripts
Editing Scripts
Using the Resource Browser
Creating a Script Palette
Using Scripts
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Scripted Plug-ins

Document scripts, as described in “Creating and Editing Script Palettes and Scripts” on page 1775, allow scripts to be created, saved, and executed from the Resource Browser of a Vectorworks file. However, scripted plug-in objects allow scripts to be available in any Vectorworks file. The three types of plug-ins—menu commands (`.vsm`), tools (`.vst`), and objects (`.vso`)—allow scripts to be incorporated into a workspace as menu commands, tool palettes, and objects in Resource Browser libraries.

Plug-in objects created with scripts become custom items that can streamline and enhance the design/drafting process. They support the standard Vectorworks program core technologies such as snapping, classing, and advanced object editing, giving them essentially the same status as Vectorworks built-in object types.

Scripted plug-ins also provide enhanced portability and platform independence for scripts, allowing them to be easily moved to Vectorworks software installations on either Mac or Windows systems. Drawings containing plug-in objects can be exchanged between users in different countries. The plug-ins can be localized for use in other countries by translating the names and strings that display.

How Plug-ins Work

Scripted plug-ins are saved as files. They combine script code with a plug-in “wrapper,” an encoded header which defines the characteristics and behaviors of the plug-in. Information such as the category of the plug-in, properties which define how the plug-in is activated, or any other information needed by the plug-in to function within the Vectorworks application framework is included within the header.

Plug-in Types

A key feature of scripted plug-ins is their smooth integration into the Vectorworks product interface. These menu commands, tools, and objects work just like any built-in Vectorworks software tool, object, or menu item. Like built-in menu commands, scripted menu commands can be set to require certain file conditions such as 2D/3D view orientation or a selected set of items in order to activate. When a menu command or tool item is selected, the script and any information needed by the plug-in is loaded into memory, and the plug-in script executes. The Vectorworks program uses information provided by the plug-in to provide the user interactions (such as snapping) and file environment for the menu command or tool to perform its defined actions. Scripted tools, like their built-in counterparts, make use of the SmartCursor and other tool-centric Vectorworks program features.

Plug-in objects have characteristics of both tools and symbols. Plug-in objects can be added to a tool palette and resemble tool items, but in use, they place instances of the object in the file much like the symbol tool places symbols in a file. Object scripts can also be invoked through events that occur in the file. Placed object instances can be modified with the Object Info palette to edit the parametric values that are used to define the object, and these changes will cause the script defining the object to execute for the object to redraw. Global file changes which force a regeneration of the file can also cause the scripts of objects placed in the file to execute. These characteristics give plug-in objects enormous flexibility in how they can be displayed within a file.

Plug-in objects can also be used in conjunction with the Resource Browser to create preconfigured object instances that need minimal editing after placement. Libraries of different object configurations based on a single plug-in object can be easily created and retrieved through the Resource Browser.

Plug-in File Location

When the Vectorworks program launches, it searches for any scripted plug-in files and registers the information necessary to activate and manage the plug-ins. Include files are searched for in the same folder where the associated plug-in file is located.

Vectorworks software searches for plug-ins in the following order and locations:

- In the user’s Plug-Ins folder, or aliases or shortcuts in the user’s Plug-Ins folder
- In the Vectorworks Plug-Ins folder and its subfolders

- Aliases or shortcuts in the Vectorworks Plug-Ins folder that point to locations outside the Vectorworks hierarchy

The user's Plug-Ins folder exists in a platform-specific location.

- **Windows XP:** C:\Documents and Settings\<Username>\Application Data\Nemetschek\Vectorworks\2015\Plug-Ins
- **Windows Vista/7/8:** C:\Users\<Username>\AppData\Roaming\Nemetschek\Vectorworks\2015\Plug-Ins
- **Mac:** /Users/<Username>/Library/Application Support/Vectorworks/2015/Plug-Ins

When plug-in files are first created, they are always placed in the user's Plug-Ins folder.

When the Vectorworks program launches, or a workspace is activated, a plug-in is available in the current session only if it is located in the Plug-Ins folder, its subfolders, user's Plug-Ins folder and subfolders, or folders referenced by aliases (Mac) or shortcuts (Windows).

If duplicate plug-in files exist in more than one location, the instance that occurs first in the folder hierarchy is the one that is used.

The flexibility of the plug-in file location provides an advantage when upgrading from a previous version of Vectorworks software. Because third-party plug-ins can be stored in a folder separate from the application, they can easily be referenced when installing the upgrade. Copying the Plug-Ins folders to the current version's folders, or referencing them with an alias or shortcut, is all that is required when upgrading.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Scripted Plug-Ins

Creating Scripted Plug-Ins

The Plug-in Manager displays three types of plug-ins:

- Custom, unencrypted plug-ins that you create
- Third-party plug-ins, purchased from developers (encrypted), and encrypted plug-ins you create
- Built-in, encrypted plug-ins that are included with the Vectorworks program

Third-party plug-ins and built-in plug-ins, which are encrypted, may allow limited editing of the plug-in parameters and definition. Options that are not available for editing appear dimmed.

Custom, unencrypted plug-ins are created and edited from the Custom Plug-ins tab of the Plug-in Manager, where you can specify the plug-in type, category, parameters, options, and code for a new custom plug-in. Plug-ins are saved as described in "Plug-in File Location" on page 1781.

To create a plug-in:

1. Select **Tools > Plug-ins > Plug-in Manager**.

   The Plug-in Manager dialog box opens.

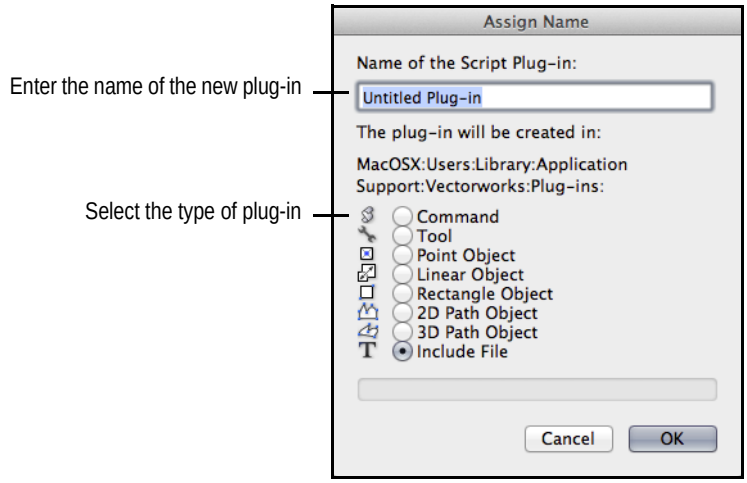
Click to show/hide the parameters.

Parameter	Description
Custom Plug-ins	
Plug-ins list	Lists available custom plug-ins; the icon to the left of the plug-in name indicates the type of plug-in (see "Plug-in Types" on page 1781)
New	Creates a new custom plug-in as described in this section
Edit Definition	Opens the Edit Plug-in Definition dialog box, to define or edit the properties of the currently selected custom plug-in (see "Plug-in Definition" on page 1785)
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Edit Script	Opens the Script Editor window, to create, rename, or edit the code that executes with the selected custom plug-in. See “Specifying the Plug-in Script” on page 1788.
Duplicate	Creates a copy of the currently selected plug-in; specify a name for the copy in the Assign Name dialog box
Delete	Deletes the currently selected plug-in; this action cannot be undone
Third-party Plug-ins	Lists any third-party plug-ins, and allows third-party plug-ins to be installed, uninstalled, and customized as permitted. • **Install**: Select the installation files from the third-party script provider; they are automatically copied to your plug-ins folder. • **Delete**: Removes the plug-in from the Vectorworks program and deletes the plug-in files. This action cannot be undone. • **Customize**: When permitted by the third-party provider, a selected plug-in may allow limited customization. Options that are not available for editing appear dimmed.
Built-in Plug-ins	Lists the plug-ins available with the Vectorworks installation; some of these allow limited customization as permitted. Options that are not available for editing appear dimmed. Examples of built-in plug-in customization are described in “Customizing Photometric Threshold Colors” on page 904 and “Customizing Size Settings for Plug-in Objects” on page 1792.

2. From the Custom Plug-ins tab, click **New**.

   The Assign Name dialog box opens. Enter the name of the new plug-in item and select the type of the plug-in to create. Plug-in names are limited to 27 characters in length. The appropriate plug-in extension will be appended to the plug-in name. Plug-ins can contain a Python script for execution.



Click to show/hide the parameters.

Parameter	Description
Name of the Script Plug-in	Specifies the name of the new plug-in; this is also the name of the plug-in file, automatically appended with the appropriate extension (`.px`, `.vsm`, `.vst`, or `.vso`)

Parameter	Description
The plug-in will be created in	Describes the location of the plug-in file; plug-ins are saved in the location indicated so that they are not overwritten when installing a new version of Vectorworks software; see “Plug-in File Location” on page 1781
Plug-in type	Select the type of plug-in to create
Command	Menu command (`.vsm`) plug-ins can be used like any standard menu command item, performing operations on the active Vectorworks file. Menu commands can detect the view state of the active Vectorworks file, or can determine if a selection set exists upon which the menu command can act.
Tool	Tool item (`.vst`) plug-ins allow scripts to be added to a Vectorworks workspace as a tool palette item. Tools make use of the SmartCursor, and can respond to file state conditions such as selection status or view orientation.
Objects	Plug-in objects (`.vso`) allow the creation of complex objects such as standard architectural or mechanical elements, “smart” drawing components like callouts or drawing borders, or other flexible objects which streamline the design process. Plug-in objects support standard Vectorworks program core technologies such as snapping, classing, and advanced object editing; they can contain up to 32,767 parameters for defining and editing the object appearance.
Point Object	Point objects are defined by a single point click for placement
Linear Object	Linear objects require a user-defined line to create the basic geometry of the object
Rectangle Object	Rectangle objects utilize a user-defined rectangle to define and create the basic geometry of the object
2D/3D Path Object	Path objects define a user-defined polygonal path or NURBS curve to create the basic geometry of the object
Include File	Specifies an additional file (`.vss` or `.px`) to be included with a script

3. Click **OK** to create the plug-in item.
4. Define the plug-in properties as described in “Specifying the Plug-in Definition” on page 1784.
5. Add the new plug-in to one or more workspaces with the Workspace Editor. See “Creating or Editing Workspaces” on page 1835. Once the item has been added to a workspace, it is available to any open file in Vectorworks without the need for importing the associated script into the active file.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Specifying the Plug-in Definition
Scripted Plug-ins

Specifying the Plug-in Definition

Unencrypted, custom plug-ins are listed on the Custom Plug-ins tab of the Plug-in Manager dialog box. Depending on the plug-in type, other settings control execution conditions, appearance, stored and default parameters, insertion options, and associate a script with the plug-in. Third-party plug-ins, encrypted plug-ins, and built-in plug-ins may allow limited editing of the plug-in definition.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Plug-in Definition
Specifying the Plug-in Script

Plug-in Definition

The parameters that define the appearance of a scripted command, tool, or object plug-in are stored in a parameter record, which is associated with each object instance placed in the file. These records store persistent data between uses and provide default parameter values. A menu command that displays a dialog box, for example, might need to store values entered by a user for later use. A tool might provide several mode options in a popup list. Should the user wish to select a different mode for the tool, the new setting can be saved and reused on a subsequent use of the tool item. These values can be stored in the parameter record of the menu command or tool and retrieved later when the command or tool is selected again. Switching files will display stored values associated with the new files or, if no parameter record exists, will display the default values of the parameter record as created by the plug-in item.

The parameters for each object instance can be modified by using the Object Info palette to access the values in the object parameter record. A default parameter record is also created when the first instance of an object (or tool) is created in the file. This default parameter record, which is distinct from the parameter records associated with object instances, stores the object default settings with the file. It is used when placing subsequent object instances to define the defaults for each new object instance.

To define the custom plug-in parameters:

1. From the Plug-in Manager Custom Plug-ins tab, select the plug-in to define or edit.
2. Click **Edit Definition**.

   The Edit Plug-in Definition dialog box opens.

Click to show/hide the parameters.

Parameter	Description
General	
Type	Displays the type of plug-in being created (menu command, tool, object, or include file)
Language	Displays either Python or VectorScript, depending on the language used for the script portion of the plug-in (see “Specifying the Plug-in Script” on page 1788)
Name	Specifies the name of the new plug-in; this is also the name of the plug-in file, automatically appended with the appropriate extension (`.px`, `.vsm`, `.vst`, or `.vso`). If renaming a plug-in, update the workspace afterwards to update the plug-in name.
Category	Assigns a plug-in to a heading category, to easily find the plug-in in the Workspace Editor
Parameters	Specifies the default parameters for the plug-in. Default parameters are applied to the plug-in at insertion. Parameters are listed in the specified order for the plug-in; change the parameter order by dragging within the # column.
Parameters list	Lists the default object parameters, alternate (localized) name if applicable, format, and default value. For linear objects, the `LineLength` parameter displays, which contains the axis length of the linear object. For rectangular objects, both the `LineLength` (the initial length of the object instance) and `BoxWidth` (the initial width of the object instance) parameters display. New default values can be specified for these parameters, but they cannot be deleted.
New	Creates a new parameter for the plug-in. The Edit Parameter dialog box opens. Specify the parameter name, alternate (localized) name if applicable, type, and default value, and click **OK** to create the plug-in parameter. To create a pop-up list of pre-defined parameter options, select the Pop-up **Type.** The Edit Choices dialog box opens. List the options; each line in the list represents a choice in the pop-up. Click **OK**. To edit the pop-up choices, click **Choices**.

Parameter	Description
Edit	Edits the selected plug-in parameter. Certain third-party and built-in plug-ins allow parameter editing.
Delete	Deletes the selected plug-in parameter
Strings	Specifies text strings used by a plug-in
Strings list	Lists the strings used by the plug-in (for example, color selector values that specify a certain color to be used by the plug-in)
Add	Adds a text string to the plug-in definition. The Add Category dialog box opens. The next available Resource ID number is assigned automatically; specify the category name and click **OK**.
Edit	Edits the currently selected string. The Edit Strings dialog box opens. Select the individual string value and click **Edit** to open the Edit String dialog box. Specify the string value and then click **OK**; the value depends on the category; for information, see http://developer.vectorworks.net.
Delete	Deletes the currently selected string
Properties	Property settings allow plug-ins to behave like standard Vectorworks menu commands, tools, and objects. These settings control behavior of the plug-in with respect to the state of the file (selection status, view orientation) as well as define the help text to display. The properties available depend on the plug-in type. Only custom plug-in properties can be specified.
Command	
Availability Options	For each drawing state, specify whether the condition is required (the drawing state condition is necessary for the command to be active), prohibited (the command is deactivated if the drawing state condition exists), or can be ignored (the drawing state does not affect the command activation)
Tooltip Help	Specifies the menu command help text; help text describing the menu command displays when the cursor pauses over the command (currently, this is only available on Mac)
Contextual Help Override	Enter a URL (starting with `http:`) to specify a web page containing help information. To specify a file relative to the Vectorworks program folder, indicate the path starting with . / (period and forward slash).
Version Created	Specifies the version number of Vectorworks when the plug-in was created. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Version Modified	Specifies the version number of Vectorworks when the plug-in was last modified. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Version Retired	Specifies the version number of Vectorworks when the plug-in was retired. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Tool	
Tool Bar String	Specifies the descriptive text to display in the Tool bar; usually this includes the name of the object, and it can include text indicating an action for the user to perform

Parameter	Description
Icon	The default icon for the plug-in can be replaced by a custom icon, if desired. With a third-party icon editor, create two versions of the icon: one for a standard-resolution display and one for a high-resolution display: • Standard-resolution: an image centered in an area 26 pixels wide by 20 pixels high. Save the icon in .png format. • High-resolution: an image centered in an area 52 pixels wide by 40 pixels high. Save the icon in .png format with the same name as the standard-resolution image with "@2x" appended to the end of the file name. For instance, if the standard-resolution icon is named 3DModelingToolSet.png, the high-resolution icon must be named 3DModelingToolSet@2x.png; otherwise the Vectorworks software will be unable to locate the icon. Click **Import PNG Icon** to import the standard-resolution icon; if a high-resolution icon is also present with the @2x designation, the software automatically imports it as well. The Vectorworks program automatically displays the appropriate image for the device's resolution.
Projection	Determines what view projection must be active. If 3D projection is required, the view is changed. If 2D projection is required, the Screen Plane is activated for the tool. Hybrid projection does not affect the view projection.
Execute Script	Tools are set by default to execute immediately when selected. In some cases, however, it may be desirable to have the script execution wait for mouse movement (such as a tool which draws interactively based on user mouse movement).
Contextual Help Override	Enter a URL (starting with `http:`) to specify a web page containing help information. To specify a file relative to the Vectorworks program folder, indicate the path starting with . / (period and forward slash).
Version Created	Specifies the version number of Vectorworks when the plug-in was created. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Version Modified	Specifies the version number of Vectorworks when the plug-in was last modified. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Version Retired	Specifies the version number of Vectorworks when the plug-in was retired. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Tooltip Help	Specifies the help text to display when the cursor pauses over the object icon in a palette
Object	
Icon	The default icon can be replaced by a custom icon, if desired. With a third-party icon editor, create an 8-bit image, centered in an area 26 pixels wide by 20 pixels high. Save the icon in .png format, and click **Import PNG Icon** to import it.
Projection	Determines what view projection must be active. If 3D projection is required, the view is changed. If 2D projection is required, the Screen Plane is activated for the tool. Hybrid projection does not affect the view projection.
Execute Script	Objects are set by default to execute immediately when selected. In some cases, however, it may be desirable to have the script execution wait for mouse movement (such as an object which draws interactively based on user mouse movement).
Default Class	Specifies the default class for the object upon insertion; if the class does not exist when the object is placed, the class is automatically created

Parameter	Description
Contextual Help Override	Enter a URL (starting with `http:`) to specify a web page containing help information. To specify a file relative to the Vectorworks program folder, indicate the path starting with . / (period and forward slash).
Version Created	Specifies the version number of Vectorworks when the plug-in was created. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Version Modified	Specifies the version number of Vectorworks when the plug-in was last modified. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Version Retired	Specifies the version number of Vectorworks when the plug-in was retired. For example, for Vectorworks 2013, 2014, and 2015, use 18, 19, and 20, respectively.
Tooltip Help	Specifies the help text to display when the cursor pauses over the object icon in a palette
Options (custom parametric object types only)	
Execution Options	Objects, like symbols, can be assigned predefined insertion options for document placement. These options allow objects to properly interact with walls or other advanced Vectorworks object types. Command and tool plug-ins do not have insertion option settings. By default, object geometry will only be recalculated if the object parameters or control points are edited. When object geometry is recalculated, file default settings for attributes such as font, text size, or line color will be reapplied to the object. If any of these settings have been modified since the object was placed or last edited, changes in the appearance of the object may occur. The script is called whenever the object parameters have changed. Additionally, for instances where it is important for the object to recalculate (for example, windows placed in a wall), the script can cause geometry to be recalculated when the object is rotated (**Reset on Rotate**) or moved (**Reset on Move**). Alternatively, the plug-in script can become an event handler (**Event-Based**); scripts must then respond to a small set of application events. For samples and documentation on event handling scripts, access the Support area at www.vectorworks.net
Insert in Walls	Select the wall insertion option settings for the object. See “Creating New Symbols” on page 239. For objects that do not require insertion options, leave the options at the default settings.

3. When the plug-in parameters have been defined, click **OK** to return to the Plug-in Manager.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Specifying the Plug-in Script
Specifying the Plug-in Definition
Scripted Plug-ins

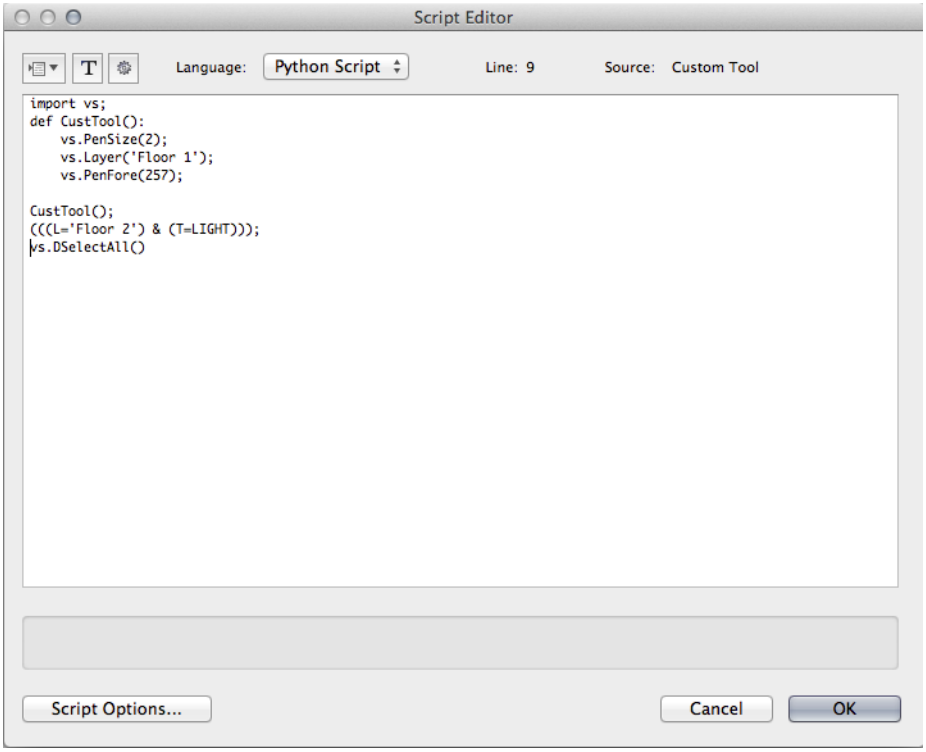
Specifying the Plug-in Script

The script source code for the command, tool, or object can be created using the Script Editor or a third-party text editor and imported into the plug-in. The source code is saved as part of the plug-in item. Either Python or VectorScript can be used for the code. For third-party or built-in plug-ins, only unencrypted plug-in scripts can be edited.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To create script code:

1. Click the **Script** button from the Plug-in Manager dialog box.

   Enter the script source code in the Script Editor window. See “Creating Scripts” on page 1776 for more information. For detailed developer-oriented documentation, please navigate to http://developer.vectorworks.net



2. Click **OK** to save the script as part of the plug-in.

   During plug-in development, scripts can be easily compiled and tested when developer mode is enabled in the Session tab of Vectorworks preferences.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Specifying the Plug-in Definition
Scripted Plug-ins

D Creating Custom Path Objects

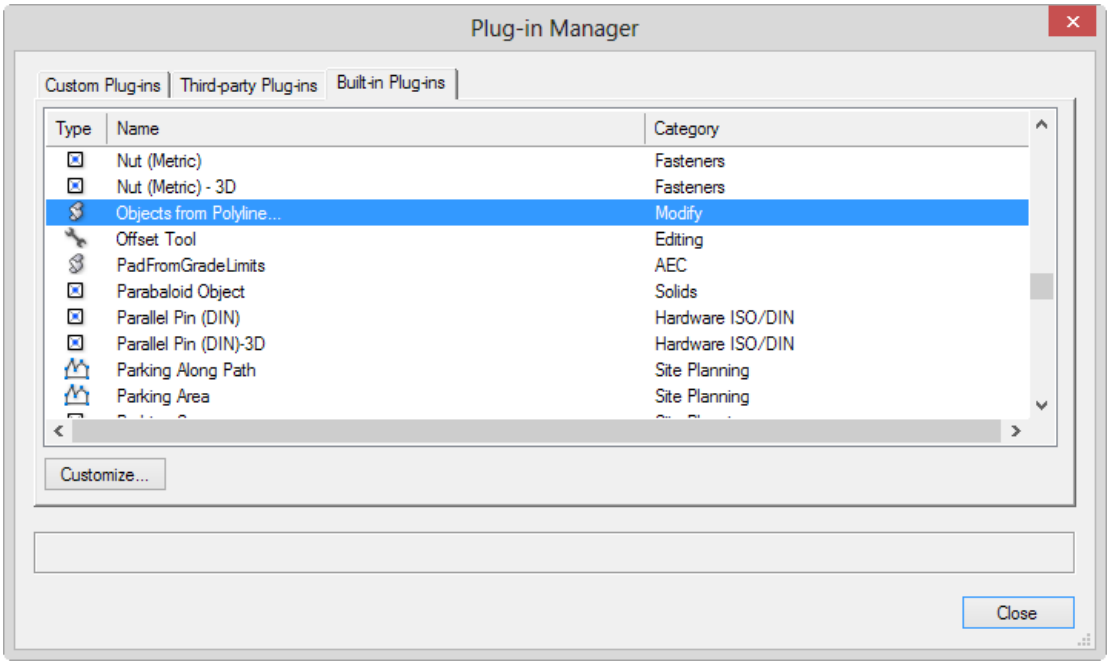
The **Create Objects from Shapes** command supports custom path objects (custom path plug-in objects with a .vso extension).

For information on creating custom path objects, see “Scripted Plug-ins” on page 1781.

To add a custom path object to the **Create Objects from Shapes** command:

1. Ensure that the custom path object file is located in the Vectorworks\Plug-Ins folder.
2. Select **Tools > Plug-ins > Plug-in Manager**.

   The Plug-in Manager dialog box opens. Click on the Built-in Plug-ins tab.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

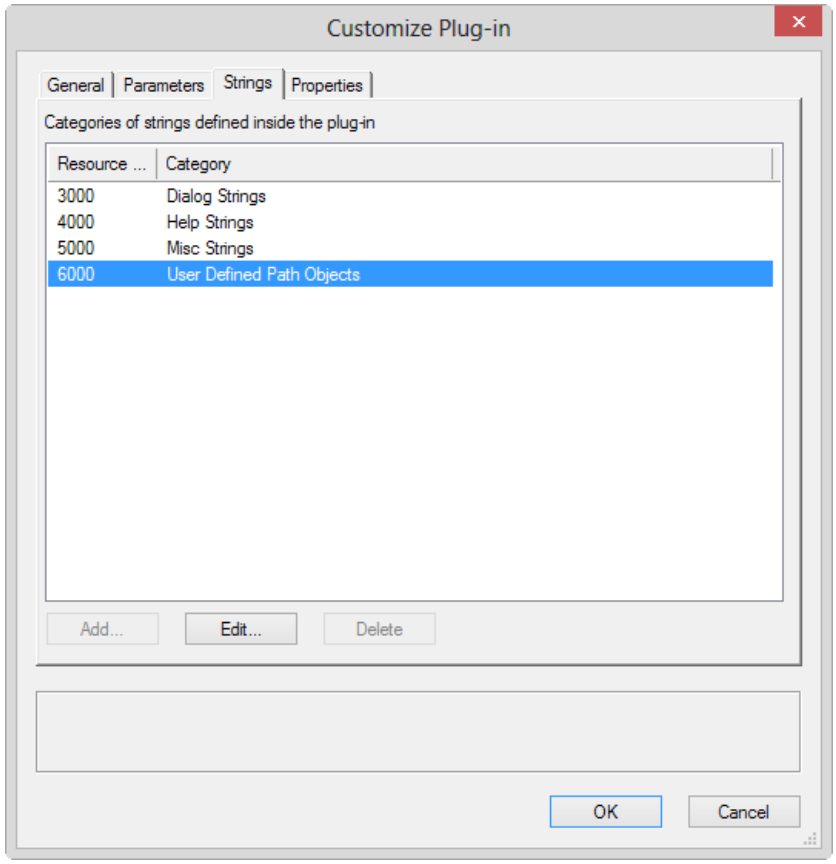


3. In the list of VectorScript plug-in objects, select Objects from Polyline.

   Press the letter O key to quickly access Objects from Polyline in the list.

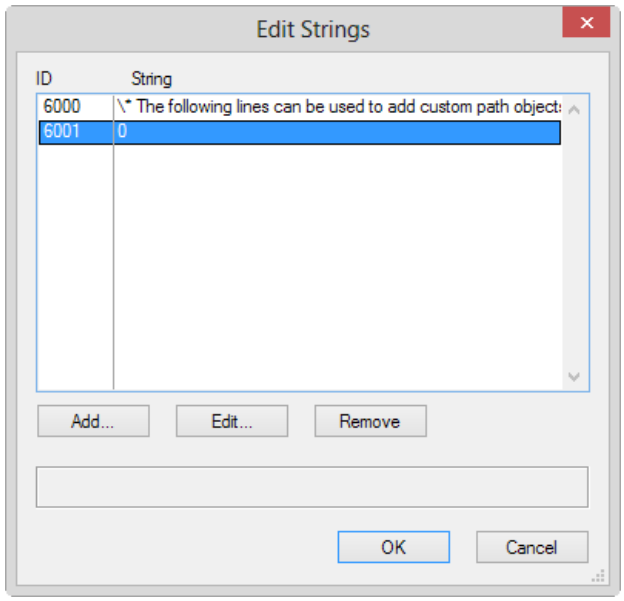
4. Click **Customize**.

   The Customize Plug-in dialog box opens. Click on the Strings tab.



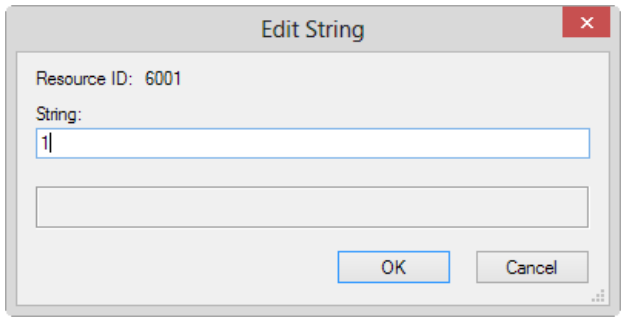
5. Select Resource ID 6000, User Defined Path Objects, and click **Edit**.

   The Edit Strings dialog box opens.

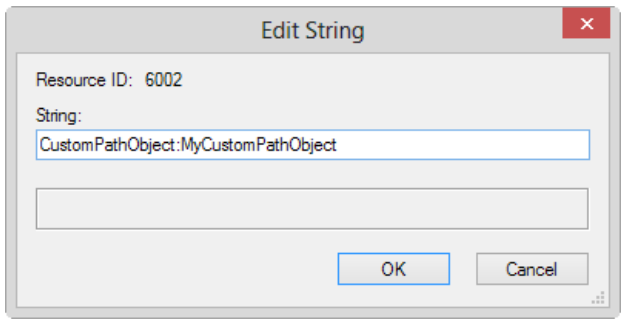


6. Select ID 6001, and click **Edit**.

   The Edit String dialog box opens. Enter the number of custom path objects that will be added, and click **OK**. (If adding one path object, enter 1.)

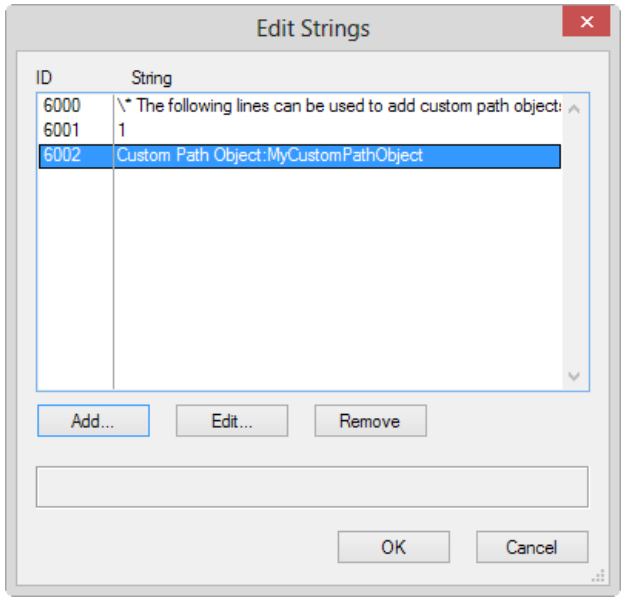


7. Click **OK**.
8. In the Edit Strings dialog box, click **Add**.
9. Enter the name of the path object that should be used in the command list, and then the actual name of the plug-in object file (without the .vso extension), separated by a colon.



10. Click **OK** to return to the Edit Strings dialog box.

    The custom path object ID and string are listed.



11. Repeat steps 8-10 for each custom path plug-in object to add.
12. Click **OK** twice, and then **Close**, to exit the Plug-in Manager.
13. The custom path object is included in the list of path objects created by the **Create Objects from Shapes** command.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Objects from Shapes

D Customizing Size Settings for Plug-in Objects

User-defined sizes can be created for eye bolts, J-bolts, swing bolts, swing eye bolts, T-bolts, U-bolts, and tapped holes. Use the following procedures to add or remove sizes for use with these plug-in objects. Sizes are then assigned through the Object Info dialog box for the applicable object.

To add or remove plug-in object sizes:

1. Select **Tools > Plug-ins > Plug-in Manager**.

   The Plug-in Manager dialog box opens. Click on the Built-in Plug-ins tab.
2. Select the plug-in object to be edited, and then click **Customize**.

   The Customize Plug-in dialog box opens. Click on the Parameters tab.
3. Select the **size** parameter and click **Edit**.

   The Edit Parameter dialog box opens.
4. Click **Choices**.

   The Edit Choices dialog box opens.
5. To remove a size, highlight it and press the Backspace key. To add a new size, insert the cursor in the desired location in the list of sizes. Enter the new parameters. To edit a size, simply type over the existing parameters. Use the following guidelines to add or edit a size:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Size	Description
Inch	Enter the size as a fraction or decimal. Enter a dash as a separator (with optional spaces before and after). Enter the threads per inch. Example: 1/2 - 20
Metric	Enter either an upper or lowercase “m.” Enter the nominal size in millimeters. Enter either an upper or lowercase “x” as a separator (with optional spaces before and after). Enter the thread pitch. Example: M8 x 1.25

Do not leave blank lines in the list of sizes. Also, if sizes are added to a 2D plug-in object that has a 3D counterpart, the same sizes must be added to the 3D plug-in object in order for the **Create 3D Object from 2D** command to work properly.

6. Click **OK** to accept the changes, and then continue to click **OK** until you can close the Plug-In Manager dialog box.

The VectorScript Debugger

If you prefer using VectorScript over Python for your scripts, VectorScript provides a powerful tool to assist in solving problems that may occur while developing scripts. This tool, known as a **source-level debugger**, controls the execution so that the internal operations of the script can be observed while the script is running. Using the debugger, it becomes possible to locate and solve problems by moving through the script line by line to view the associated data, variables, and flow of script execution.

For a detailed description of the VectorScript debugger and VectorScript, please navigate to the detailed developer-oriented documentation at http://developer.vectorworks.net

Database Connectivity

ODBC Workflow

The Vectorworks Design Series products include the ability to communicate with external databases according to Open Database Connectivity (ODBC) standard. Using ODBC, a set of data can be used to automatically update the records attached to objects. Similarly, Vectorworks objects can send object data to a database, updating it. To successfully use ODBC in Vectorworks Design Series products, a database management system is required, and steps need to be followed to set up the connection and manage the data flow.

1. Configure and set up data in the database program of your choice. Database connectivity in Vectorworks software is supported for the following database systems:
   - Microsoft Excel
   - FileMaker Pro
   - Microsoft Access
   - MySQL
   - SQLite*
   - Any database that supports ODBC and has either Mac or Windows drivers

   Database and/or operating systems typically include a separate ODBC Administrator or Manager application, which defines drivers and data sources. Database drivers may need to be installed. See “ODBC Driver Information” on page 1818.

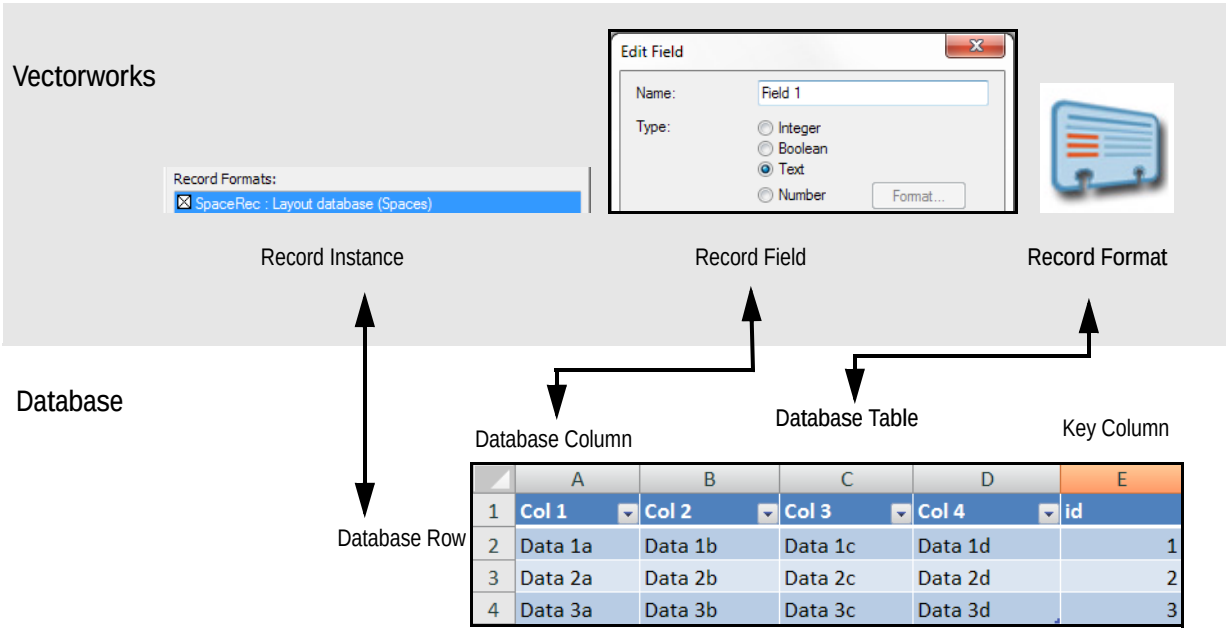
   The installation, setup, and usage of database systems is beyond the scope of this help system and is not described here. Please consult the documentation related to your operating system or database software for information.

   * SQLite database files do not require an ODBC manager or server. For information on SQLite, visit http://www.sqlite.org
2. Set up the connection between Vectorworks software and the database system. A “key” column with unique values is designated, to establish and maintain the communication between the database system and Vectorworks record formats.
3. Identify the object instances of each type of object to be connected, and define the link between the database and the record formats of the object types.
4. When connections have been established, determine the settings to update the Vectorworks record formats from the database, and to update the database from the Vectorworks object records. The two-way communication process can be automated.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Database-Vectorworks Communication

The values in a database link to the record fields of an object instance through a series of connections made between the database software and the Vectorworks software.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



One or more columns of the database table must have unique values. These Key columns are used by the Vectorworks program to uniquely identify the database rows.

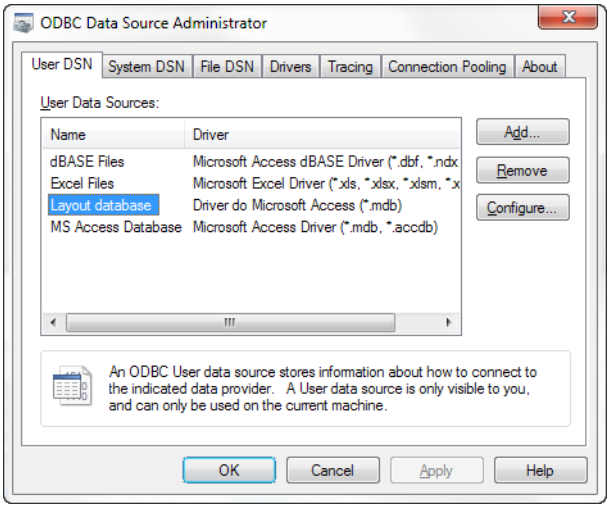
Database setup consists of identifying the key column(s) and setting up each type of connection, and then specifying update settings for the flow of data.

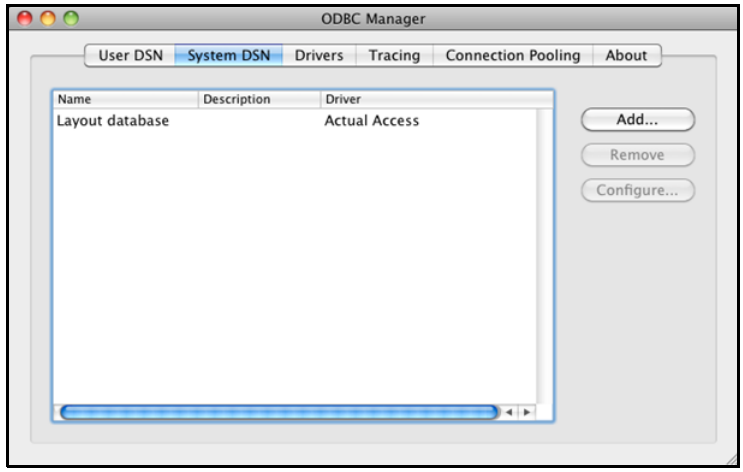
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Database Setup

In Vectorworks Design Series products, open the ODBC Data Source Administrator (Windows) or ODBC Manager (Mac). The standard drivers may already be installed for the major database providers; verify this on the Drivers tab. More information on drivers is available in “ODBC Driver Information” on page 1818. Add the database file, providing a name for it. This makes the database available through the ODBC administrator.

SQLite database files do not require an ODBC manager, so setting up the database drivers is not necessary.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~





~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Database Connection
Record Format Database Connection
Object Database Connection
Specifying Update Settings
ODBC Workflow
Database-Vectorworks Communication

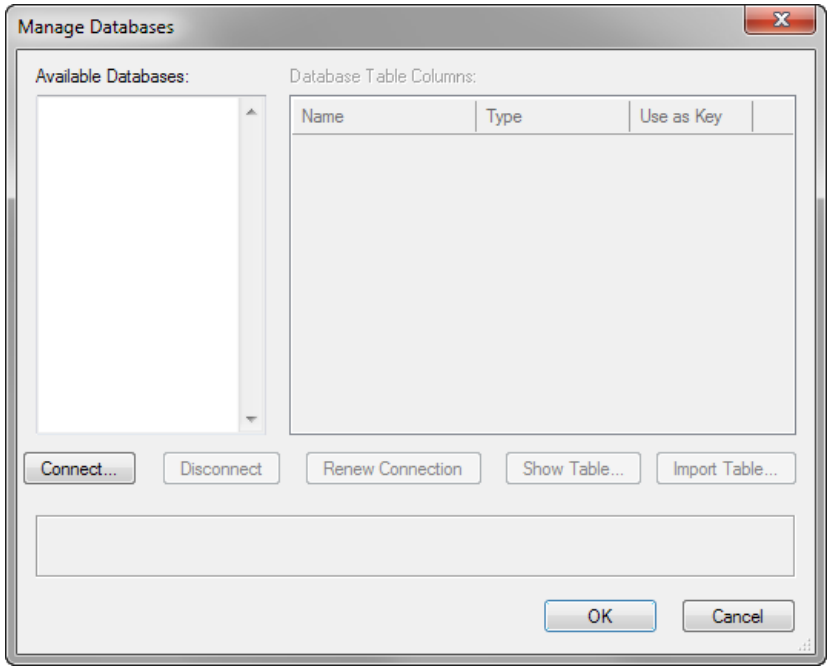
Database Connection

In the Vectorworks file, connect the database to the file and then select a Key identifier for each data source.

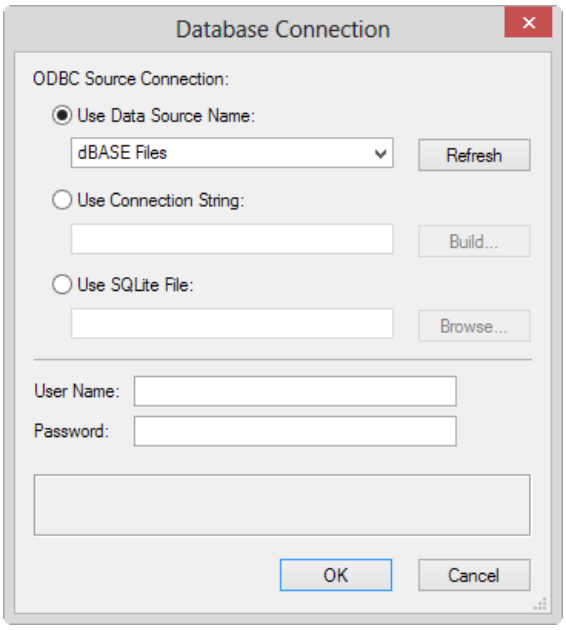
To establish the connection between the database and the Vectorworks file:

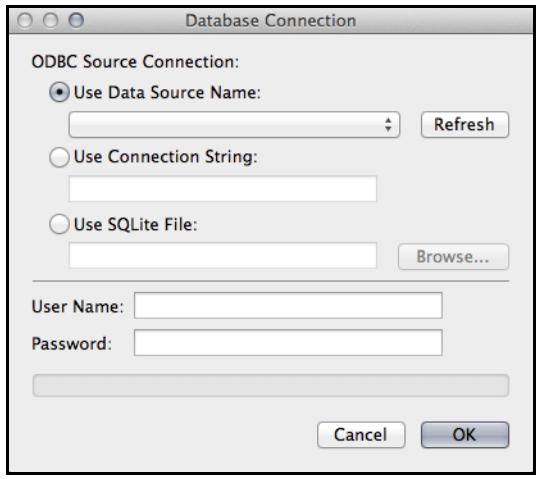
1. Open the Vectorworks drawing. Database connection information is saved in the Vectorworks file.
2. Select **Tools > Database > Manage Databases**.

   The Manage Databases dialog box opens. Through this dialog box, manage available databases and connect the desired database or databases to the current file.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



3. Since the connection has not yet been made, the **Available Databases** list is empty. Click **Connect**.
   The Database Connection dialog box opens.



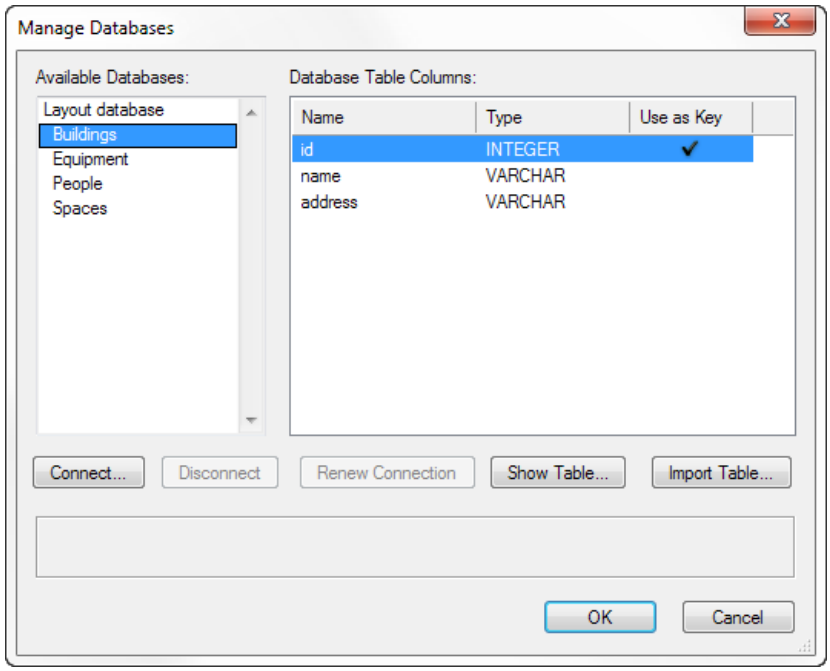


Click to show/hide the parameters.

Parameter	Description
Use Data Source Name	Connect to a database source that was previously added to the ODBC manager or ODBC administrator; click **Refresh** to update the choices

Parameter	Description
Use Connection String	On Windows, this option allows temporary creation of data sources from files, or can select the known data source name similarly to the Use Data Source Name option. Click **Build** and select the database from the Machine Data Source tab. The connection string cannot be built from a specified source on the Mac.
Use SQLite File	Click **Browse** to select and connect to an SQLite database file; this type of database connection is indicated as [sqlite] in database-related dialog boxes. If no SQLite file exists, browse to an appropriate location and enter a file name to automatically create an empty SQLite database file.
User Name/Password	If a database connection requires a user name/password, enter the information

4. Either select the SQLite file, or the named database added to the ODBC manager/administrator earlier, enter a **User Name** and **Password** if required, and click **OK**.
5. The selected database is added to the list of available databases in the Manage Databases dialog box. The tables associated with the Layout database are listed on the left, with the database table columns listed on the right.



6. Select one of the available database tables; its rows display on the right. Select one of the rows to serve as the Key identifier by clicking in the **Use as Key** column for that row. The Key column uniquely identifies the data rows in the table. By default, this is one of the columns with unique data, used as a row identifier.

   The ability to select the Key depends on the database driver. If needed, click **Show Table** to display the table's contents and help select an appropriate Key.
7. Select the Key identifier for any other database tables. Each table must have at least one column selected as **Use as Key**.
8. Click **Connect** if there are other databases to connect, and identify the Key for each additional table.
9. Once the database tables have been selected and connected, and the Key identifier(s) set for each table, click **OK**.

   The data sources have now been properly connected and identified for the file.

If the connection to the database becomes out of date, select the database and click **Renew Connection** to re-establish the database connection. Table and column information data is updated.

If there are problems connecting to the database, an error message provides troubleshooting information. Click **Details** to open a log file in the default text editor application. The log file provides further information to help diagnose connection errors.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Database Setup
Record Format Database Connection
Object Database Connection
Specifying Update Settings
Database-Vectorworks Communication

Record Format Database Connection

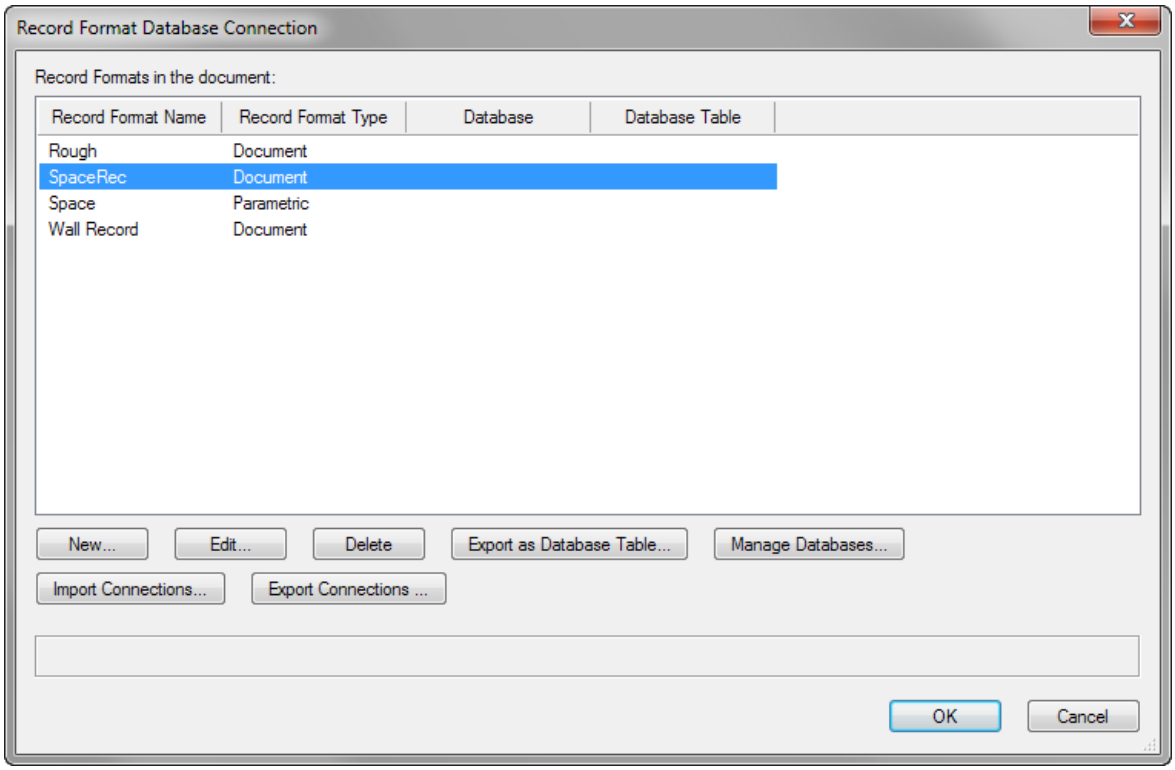
Each field of the record format needs to be mapped to the appropriate database table column. The record connections can be saved in an xml file and imported into another file.

Record formats can be automatically created and linked to the database table; see “Automatically Creating Record Formats from a Database Table” on page 1811.

To connect the desired record format to the database table:

1. Select **Tools > Database > Record Format Connection**. Alternatively, select a record format in the Data tab of the Object Info palette for a selected object with the desired record attached, and then click **Record Format Database Connection**.

   The Record Format Database Connection dialog box opens, showing available record formats and parametric objects in the file. Each record format can be linked to a data source that has already been connected to the file.

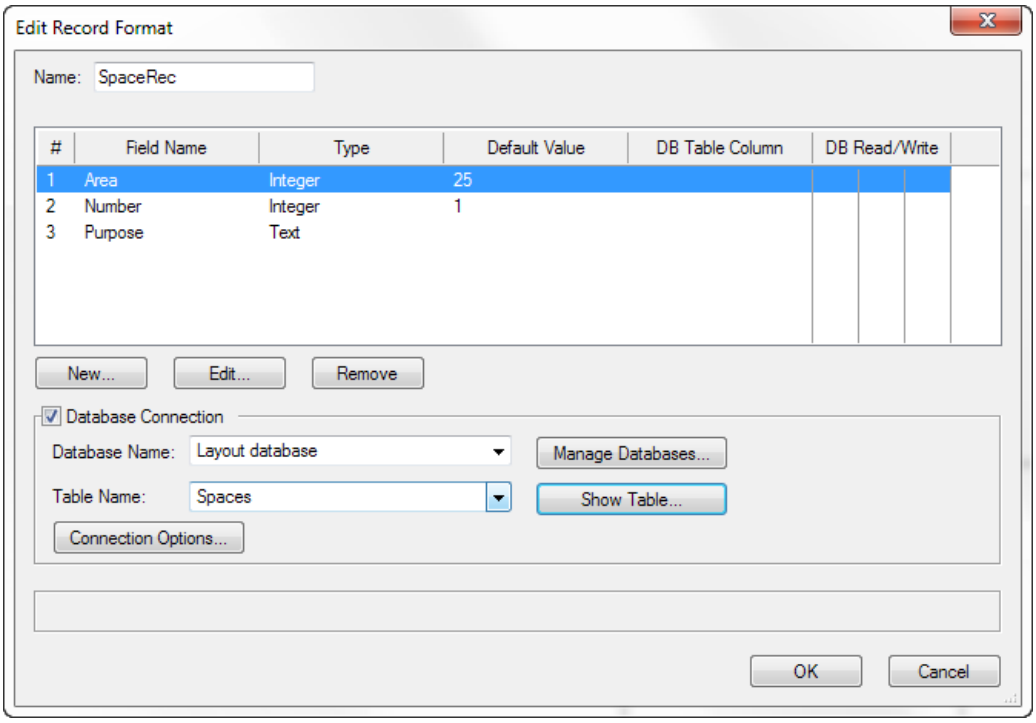


Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Record Formats in the Document	Lists the record formats available in the Vectorworks file, as well as the databases and database tables to which they are connected
New	Creates a new record format as described in “Creating Record Formats” on page 262
Edit	Opens the Edit Record Format dialog box, to link the record format to the specified database and table
Delete	Deletes the selected record format from the file
Export as Database Table	Automatically creates database table rows from the selected record format; see “Automatically Creating a Database Table from a Record Format” on page 1813
Manage Databases	Opens the Manage Databases dialog box, to connect to available databases; see “Database Connection” on page 1797
Import Connections	Imports the connection settings from an external .xml file, overriding any existing database connections. This allows the connections specified from another file to be used in this file.
Export Connections	Exports the current connection settings to an .xml file

2. Select the record format and click **Edit**.

   The Edit Record Format dialog box opens, with additional database controls available due to the presence of the database connection.



3. Select **Database Connection** to link the record format fields to the selected database and table. This area only displays for record formats when a database is connected in the file.

   If needed, verify or change the connected database by clicking Manage Databases, or check the table selected in **Table Name** to view the table contents.

4. Click **Connection Options** to specify database connection update options.

The Connection Options dialog box opens. Select the actions to occur in the database table when the connected record format, or object with the record attached, is edited. (These changes occur at the time of database update set in “Specifying Update Settings” on page 1817; alerts inform you of any changes to the database.)

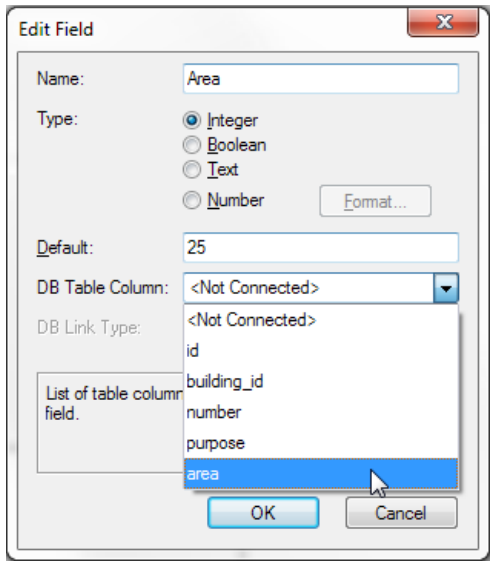
5. Click **OK** to return to the Edit Record Format dialog box.

Click to show/hide the parameters.

Parameter	Description
Add new rows in Database Table when creating Records of this Format	When a new record format is created, automatically adds a database table row to the connected database
Delete the row in the Database Table when removing Records of this Format	When a record format is disconnected from an object, or an object with a connected record format is deleted, automatically deletes the row that was connected to the object in the database table

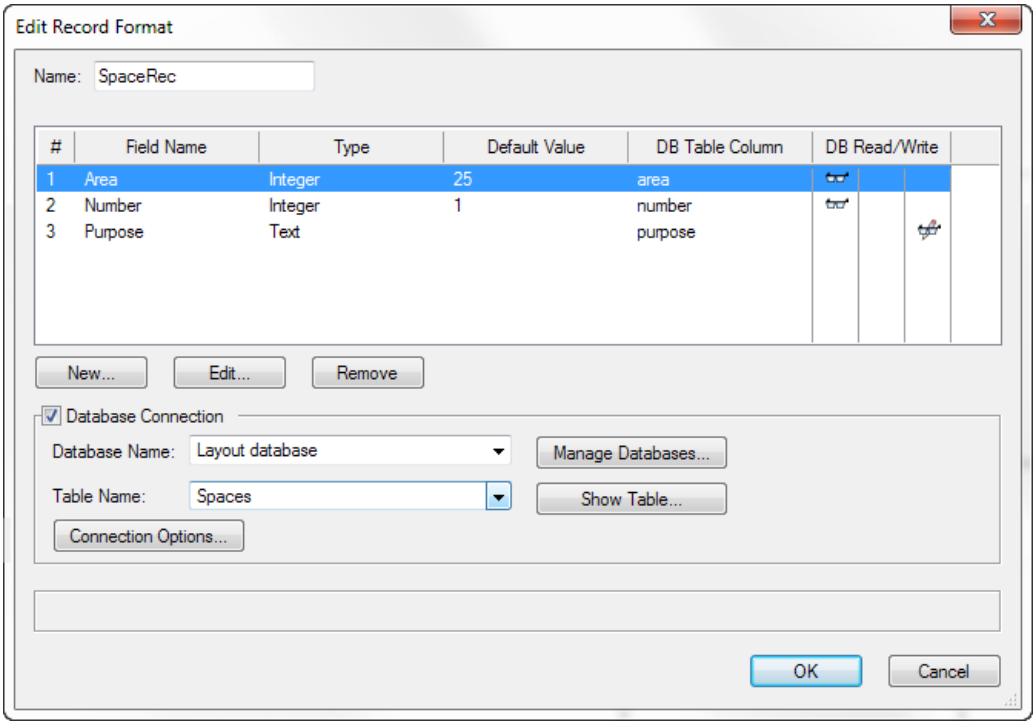
6. For each record field, assign a database table from the selected database. Select a record field and click **Edit**.

   The Edit Field dialog box opens.

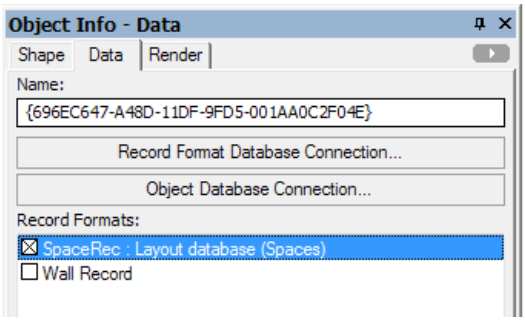


7. Select the associated table row from the database in **DB Table Column**, and select the method of database communication in **DB Link Type**:
   - **Read/Write**: Allows data to move in both directions between the database and the Vectorworks file
   - **Read Only**: Allows database information to be read into the Vectorworks file, but does not allow record data to modify the database
   - **Write Only**: Allows the Vectorworks file data to modify the database, but does not allow the database information to modify the Vectorworks record field
8. Click **OK** to return to the Edit Record Format dialog box. The dialog box updates to display the associated table data and link type.

   Click in a **DB Read/Write** column to easily change the link type for the selected row.



9. Once the data has been linked for each record field, click **OK**. The fields in the record format have now been linked to the table data in the specified database, and the link type has been defined. On the Data tab of the Object Info palette, connected record formats display with the name of the connected database and its table.



~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Database Setup
Database Connection
Object Database Connection
Specifying Update Settings
Record Formats
Database-Vectorworks Communication

Object Database Connection

The final step in setting up a database connection is to connect object instances to the database. Once connected, the connection can be edited. The connection can also be removed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Single Object Connection
Multiple Object Connection
Editing Object Connections

Removing Object Connections
Database Setup
Database Connection

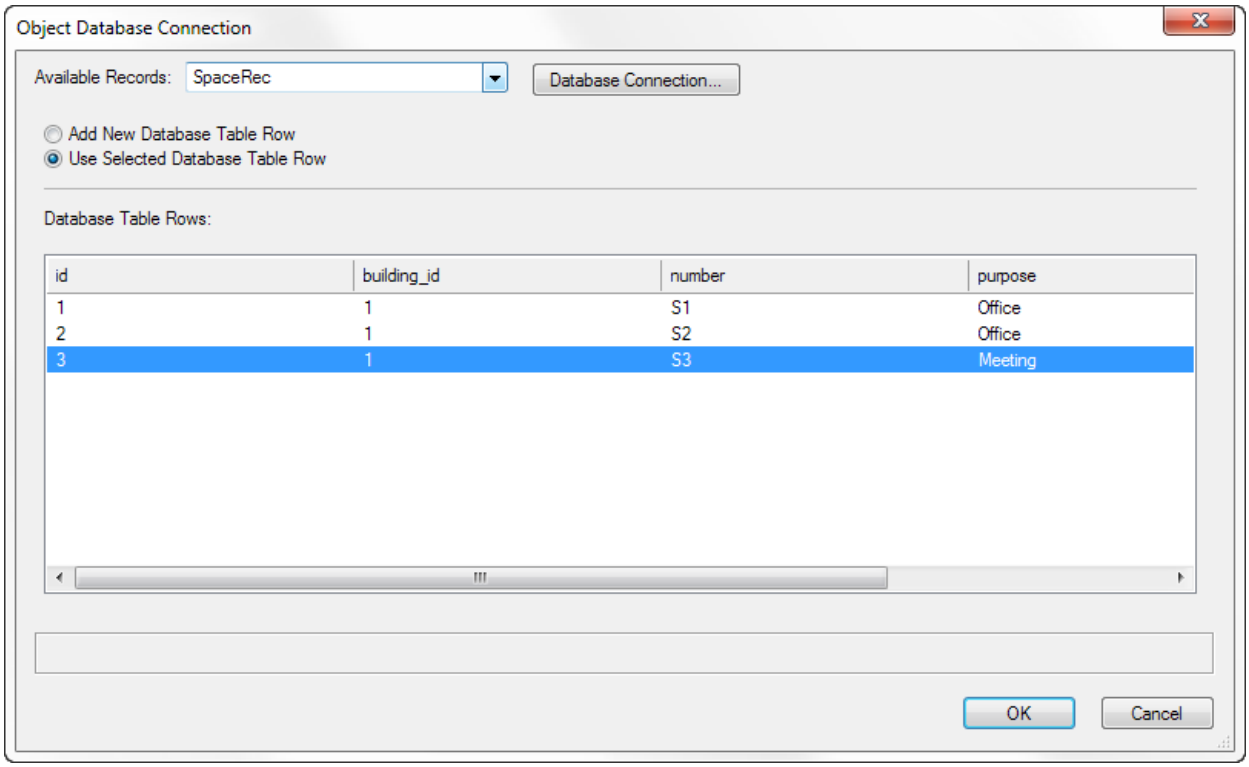
Single Object Connection

To connect a database to an object:

1. Select an object.
2. On the Data tab of the Object Info palette, click the box for the selected format to attach the record format to the object instance. (See “Attaching Record Formats to Symbols and Objects” on page 263.)

   The Record Fields display with default values from the record format.
3. Select **Tools > Database > Object Connection**. Alternatively, click **Object Database Connection** from the Data tab of the Object Info palette.

   The Object Database Connection dialog box opens. If a row was previously selected for the object, it appears highlighted and bolded.



Click to show/hide the parameters.

Parameter	Description
Available Records	Lists the record formats available for the selected object
Database Connection	Opens the Record Formats Database Connection dialog box, to connect a record format to a database table
Add New Database Table Row	Inserts a new row in the database table, and associates it with the object

Parameter	Description
Use Selected Database Table Row	Allows the database table row from the linked database table list to be selected, associating an existing row with the object
Database Table Rows	Selects a table row to associate with the object; this row provides data for the selected object or receives data from the selected object

4. Click **Use Selected Database Table Row**, and then select the appropriate row for the selected object.
5. Click **OK**.
6. The record fields are populated from the database table for the record instance attached to the object.

Multiple Object Connection
Editing Object Connections
Removing Object Connections
Database Setup
Database Connection
Record Format Database Connection
Specifying Update Settings
Record Formats
Database-Vectorworks Communication

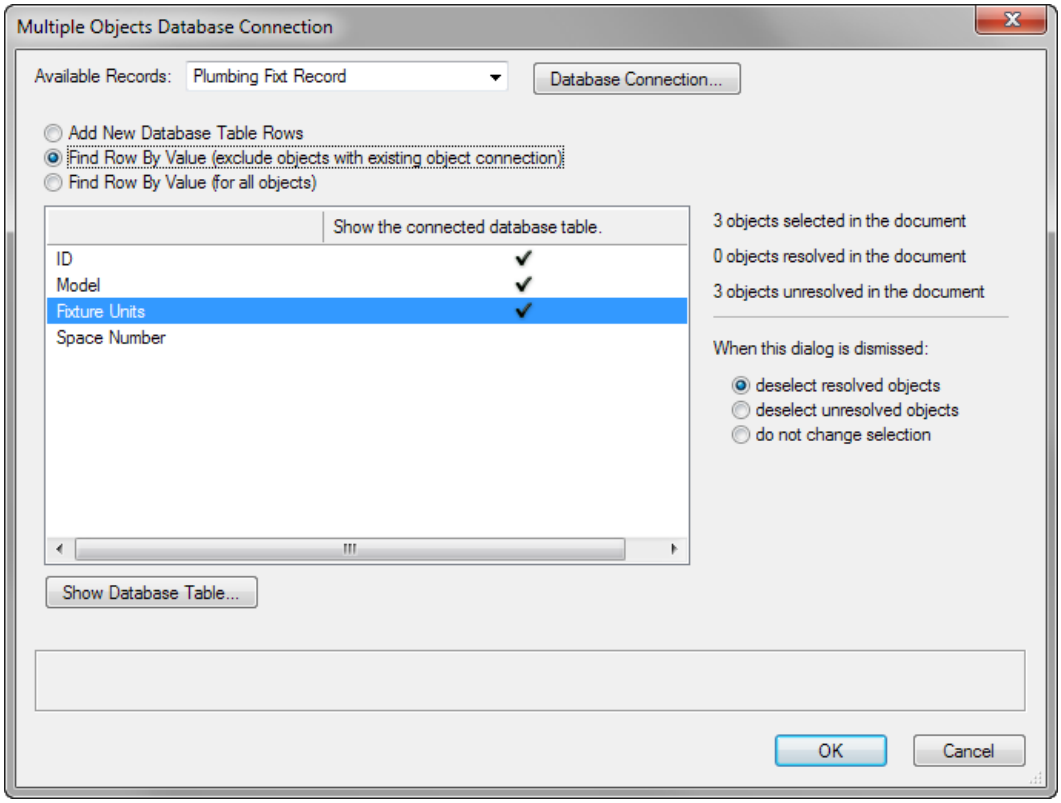
Multiple Object Connection

When there are multiple similar objects that should be identically connected to database tables through attached record formats, it would be tedious to connect each one. The connection for multiple objects can be made at one time.

To connect a database to multiple objects:

1. Select several objects of the same type; for example, select several space objects.
2. Select **Tools > Database > Object Connection**. Alternatively, click **Object Database Connection** from the Data tab of the Object Info palette.

   The Multiple Objects Database Connection dialog box opens. By selecting a format field and selecting search filter criteria, the table rows associated can be searched.



Click to show/hide the parameters.

Parameter	Description
Available Records	Lists the record formats available for the selected objects (only record formats that are common to the selected objects display)
Database Connection	Opens the Record Formats Database Connection dialog box, to connect a record format to a database table
Add New Database Table Rows	Inserts a new row in the database table for each selected object, and associates each row with each object
Find Row by Value (exclude objects with existing object connection)	Only the format fields connected to a database are listed. Selects a format field, and uses the value from the selected objects that are not already connected to a database, to search the database table for a particular row. (Once the row is found, the Key identifier is used to link the table row to the record fields.)
Find Row by Value (for all objects)	Only the format fields connected to a database are listed. Selects a format field, and uses the value from the selected objects to search the database table for a particular row. (Once the row is found, the Key identifier is used to link the table row to the record fields.)
Show Database Table	Opens the Database Table dialog box, to view the database table associated with the record format and help determine which column to use for the search criteria
When this dialog is dismissed	Determines the selection status of resolved objects, or of unresolved objects that may require further attention to connect them. Select **do not change selection** to make no change to the number of selected objects.

3. Select a **Format Field**, and then click in the **Find by Value** column to select the field as a search filter.

4. On the right, the status of resolved objects indicates how many of the selected objects had data found in the database tables based on this search. When objects are resolved, their record field data is properly connected to the database table.
5. Click **OK**. For each resolved object, the record fields are populated from the database table for the record instance attached to the objects.

   Resolved objects are connected to the corresponding rows using the Key identifier, and then each row is associated with the corresponding object.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Single Object Connection
Editing Object Connections
Removing Object Connections
Database Setup
Database Connection
Record Format Database Connection
Specifying Update Settings
Record Formats
Database-Vectorworks Communication

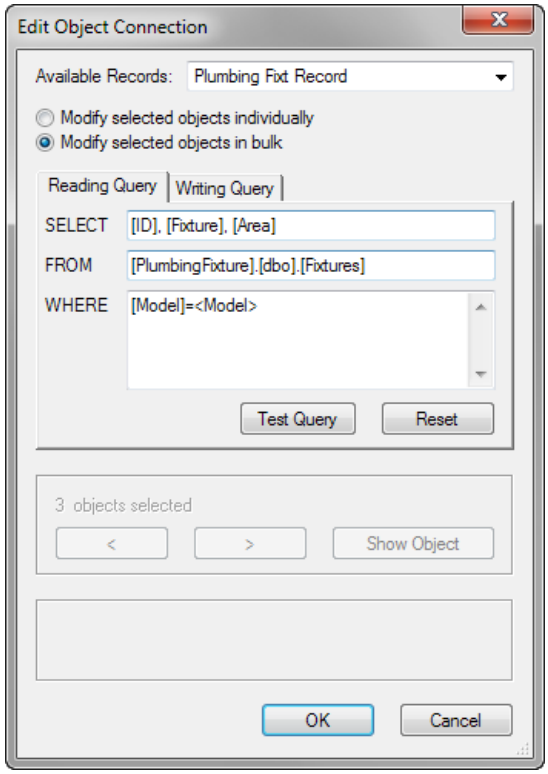
Editing Object Connections

The read/write queries of an object connection can be viewed and edited for one or several selected objects.

To edit object connections:

1. Select one or more objects. The objects should already be connected to a database.
2. Select **Tools > Database > Edit Object Connection**.

   The Edit Object Connection dialog box opens. The reading and writing SQL queries for the selected object(s) display and can be edited, either individually for each selected object, or for all the objects at once.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Click to show/hide the parameters.

Parameter	Description
Available Records	Lists the record formats available for the selected objects (when multiple objects are selected, only record formats that are common to the selected objects display)
Modify selected objects individually	Edits the reading and writing SQL queries of each selected object individually, by scrolling through each one
Modify selected objects in bulk	Edits the reading and writing SQL queries of all the selected objects at one time using record name identifiers; scrolling through the selected objects is not necessary
Reading Query	Shows the reading SQL query for the selected object(s)

Parameter	Description
Protocol	Reading SQL queries consist of the following rows: • SELECT: indicates the column names, in brackets separated by commas Example: [id],[room name],[area] • FROM: indicates the name of the database and table, in brackets separated by periods Example: [SpaceInfo].[dbo].[Spaces] • WHERE: indicates the name and value of the record format, with the record format name in brackets and the value (or identifier in angle brackets) separated by an equal sign Example: [area]=30 or [area]=<area> When editing a single query that applies to multiple selected objects (**Modify selected objects in bulk** is selected), use an identifier within angle brackets for the WHERE record field name, instead of the actual value. Using an identifier in angle brackets allows the query to proceed for all selected objects regardless of each object's actual value.
Test Query	Tests the current reading SQL query to see if it is valid; this is useful for verifying queries before committing to the changes. An alert indicates whether the test query was successful or invalid
Reset	If the reading SQL query has been modified, returns the query to its original settings
Writing Query	Shows the writing SQL query for the selected object(s)
Protocol	Writing SQL queries consist of the following rows: • UPDATE: indicates the name of the database and table, in brackets separated by periods Example: [SpaceInfo].[dbo].[Spaces] • SET: indicates the name and value of the table column, with the column name in brackets and the value (or record format field name in angle brackets) separated by an equal sign Example: [Room Name]=Kitchen or [Room Name]=<name> • WHERE: indicates the name and value of a search condition, such as column name and value, with the search item name in brackets and the value (or record format field name in angle brackets) separated by an equal sign Example: [id]=1 or [id]=<identifier> When editing a single query that applies to multiple selected objects (**Modify selected objects in bulk** is selected), use a record format field name within angle brackets for the SET and/or WHERE value, instead of the actual value. Using a record format field name in angle brackets allows the query to proceed for all selected objects regardless of each object's actual value.
Test Query	Tests the current writing SQL query to see if it is valid; this is useful for verifying queries before committing to the changes. An alert indicates whether the test query was successful or invalid.
Reset	If the writing SQL query has been modified, returns the query to its original settings

Parameter	Description
< or >	When multiple objects are selected with the **Modify selected objects individually** option enabled, scrolls among the objects to display each object's queries for editing
Show Object	Displays the current object in the drawing window for identification

3. View or edit the object connection information, and then click **OK**.

   If there are validation problems, an error message provides information. The cursor is automatically placed in the field where the problem exists to help identify the problem.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Single Object Connection
Multiple Object Connection
Removing Object Connections
Database Setup
Database Connection
Record Format Database Connection
Specifying Update Settings
Record Formats
Database-Vectorworks Communication

Removing Object Connections

The connection between an object and a database can be removed.

To remove object connections:

1. Select one or more objects. The objects should already be connected to a database.
2. Select **Tools > Database > Remove Object Connection**.

   The database is disconnected from the object(s).

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Single Object Connection
Multiple Object Connection
Editing Object Connections
Database Setup
Database Connection
Record Format Database Connection
Specifying Update Settings
Record Formats
Database-Vectorworks Communication

Automating Database Connection Workflows

In the Vectorworks Design Series, the functionality for connecting to a database and then establishing object/record format database linking contains built-in automation features to make the job easier. While it is still necessary to know how to accomplish the steps manually, dynamic updating between the Vectorworks file and the connected database allows record formats to be automatically created from a database table, and database tables to be automatically created from record formats. Connections are automatically established, and remain up to date.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Automatically Creating Record Formats from a Database Table
Automatically Creating a Database Table from a Record Format
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

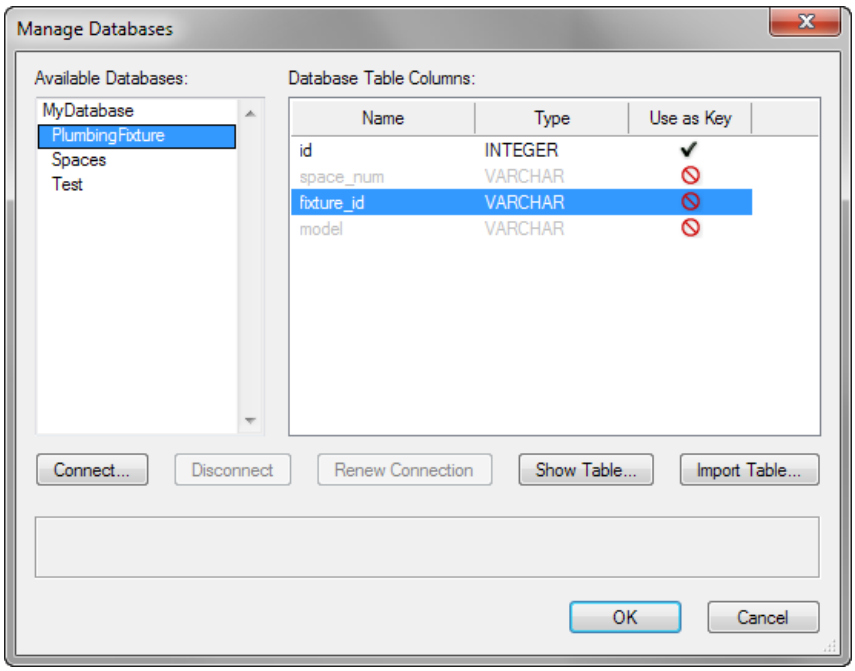
D Automatically Creating Record Formats from a Database Table

Record formats can be created from database tables. The imported tables are automatically linked to the new record formats.

To create record formats from a table in a connected database:

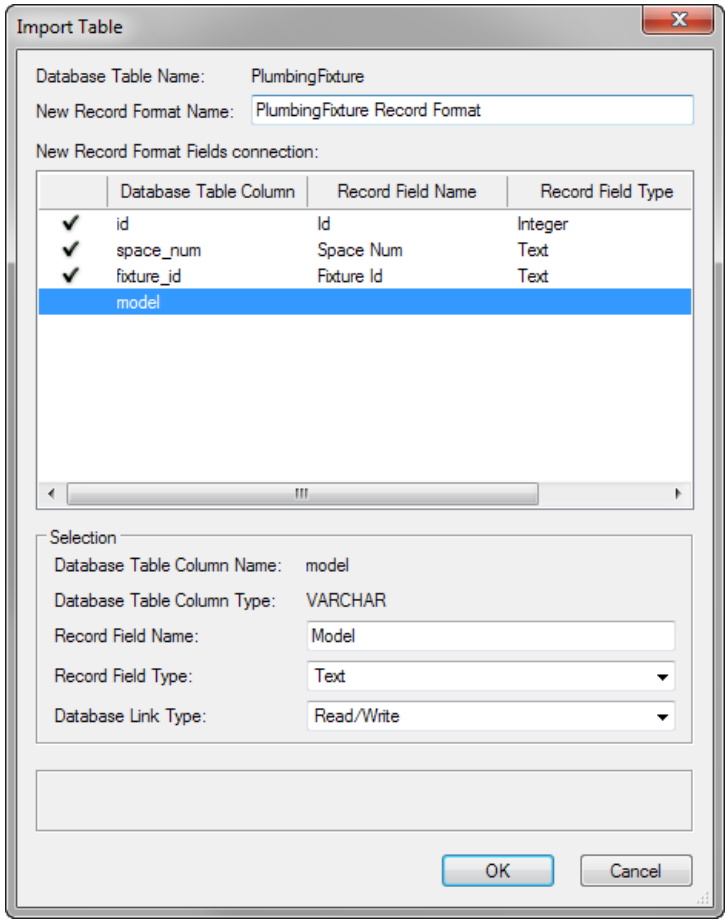
1. Select **Tools > Database > Manage Databases**.

   The Manage Databases dialog box opens.



2. Select the database on the left, and then click **Import Table.**

   The Import Table dialog box opens.



Click to show/hide the parameters.

Parameter	Description
Database Table Name	Displays the name of the selected database
New Record Format Name	Specifies the name of the new record format to create; the suggested name is based on the database table name, but it can be changed
New Record Format Fields connection	Lists the database columns and the corresponding record field and type; database fields with a check mark in the left column become record fields in the Vectorworks file
Selection	Specifies the parameters and link type for the selected record field to be created; the **Record Field Name**, **Record Field Type**, and **Database Link Type** are suggested based on the column information, but they can be changed

3. Select the columns from the database that will become record fields in the new record format by placing a check mark in the left column. For each selected column, specify the record field name, type, and connection type. For more information on connection types, see “Record Format Database Connection” on page 1800.
4. Click **OK**.

   Each selected table column in the database table becomes a record field in the new record format. The record fields are automatically connected to the database according to the connection type specified.

Automatically Creating a Database Table from a Record Format

Specifying Update Settings
Record Format Database Connection

D Automatically Creating a Database Table from a Record Format

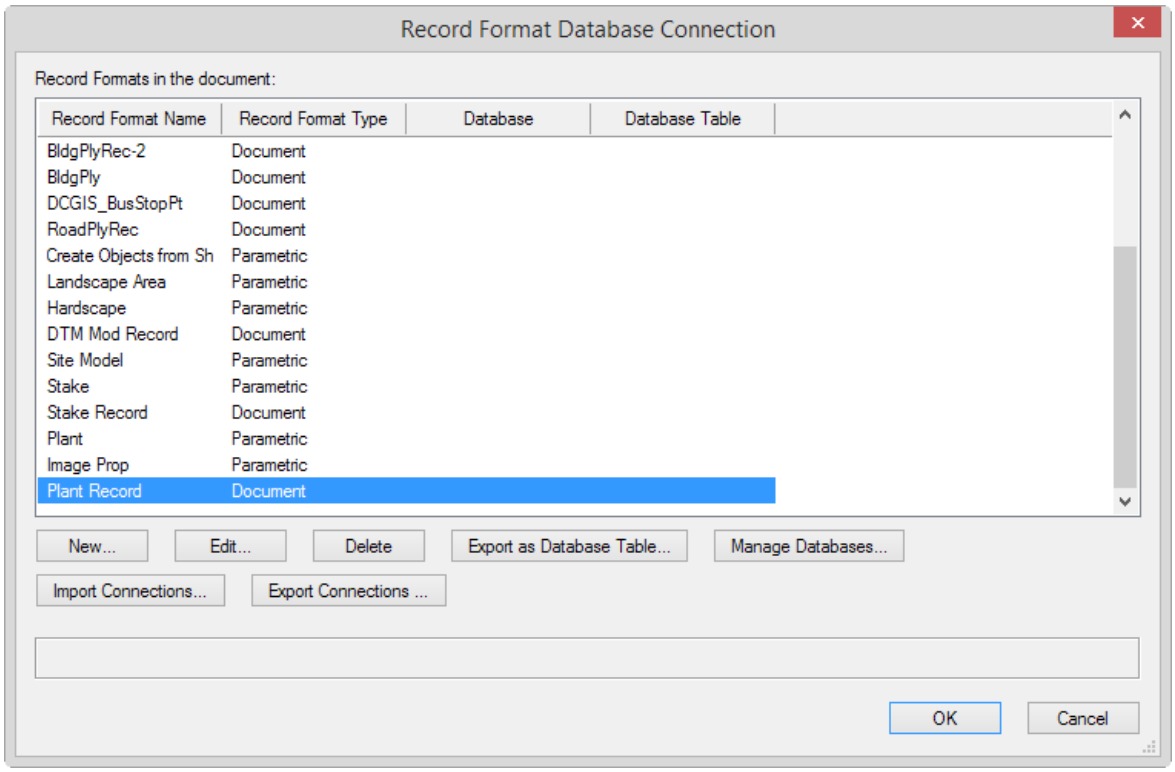
Database tables can be created from record formats. These steps quickly create a database table and link objects to it:

- Connect to a database as described in “Database Connection” on page 1797. Connecting to an empty sql database is the easiest method.
- Select the record format and create the database rows as described in the following steps.
- Connect the objects with that record format to the database table as described in the following steps. The new tables are automatically linked to the record formats.

To create a database table from a record format:

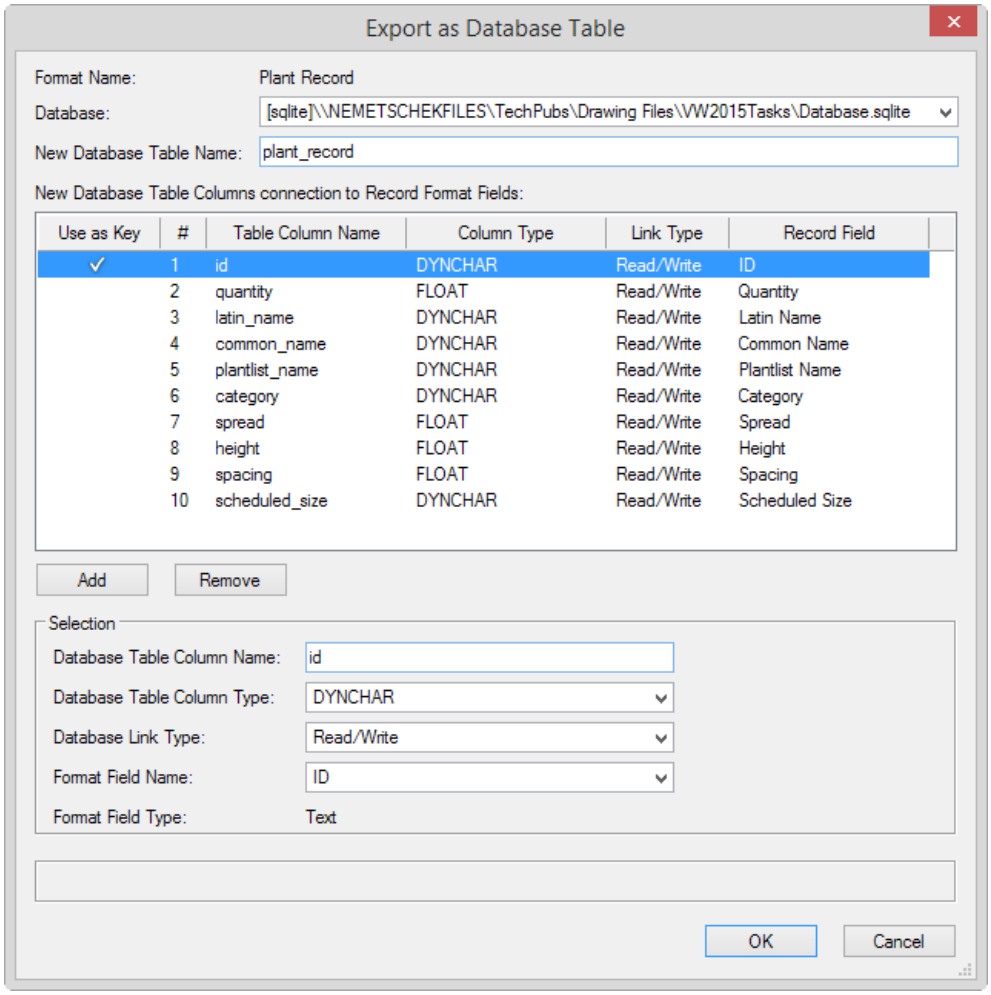
1. Select **Tools > Database > Record Format Connection.**

   The Record Format Database Connection dialog box opens, listing the record formats available in the current file.



2. Select the record format, and then click **Export as Database Table**.

   The Export as Database Table dialog box opens.



Click to show/hide the parameters.

Parameter	Description
Format Name	Displays the name of the selected record format
Database	Lists the databases connected to the current file; select the database to receive the new table
New Database Table Name	Specifies the name of the new data table to create; the suggested name is based on the record format name, but it can be changed
New Database Table Columns connection to Record Format Fields	Lists the columns and rows, based on the record format fields, that will become database table columns; the table column names are based on the record fields, but they can be changed in **Selection.** Remove any record format rows for data that does not need to be connected to a database.
Key	Selects which field from the record format is used to match the object records to the particular rows in the database. A key must be selected.
Add	Adds table rows; select each added row and specify its properties in **Selection**. This action does not affect the existing record format in the file.

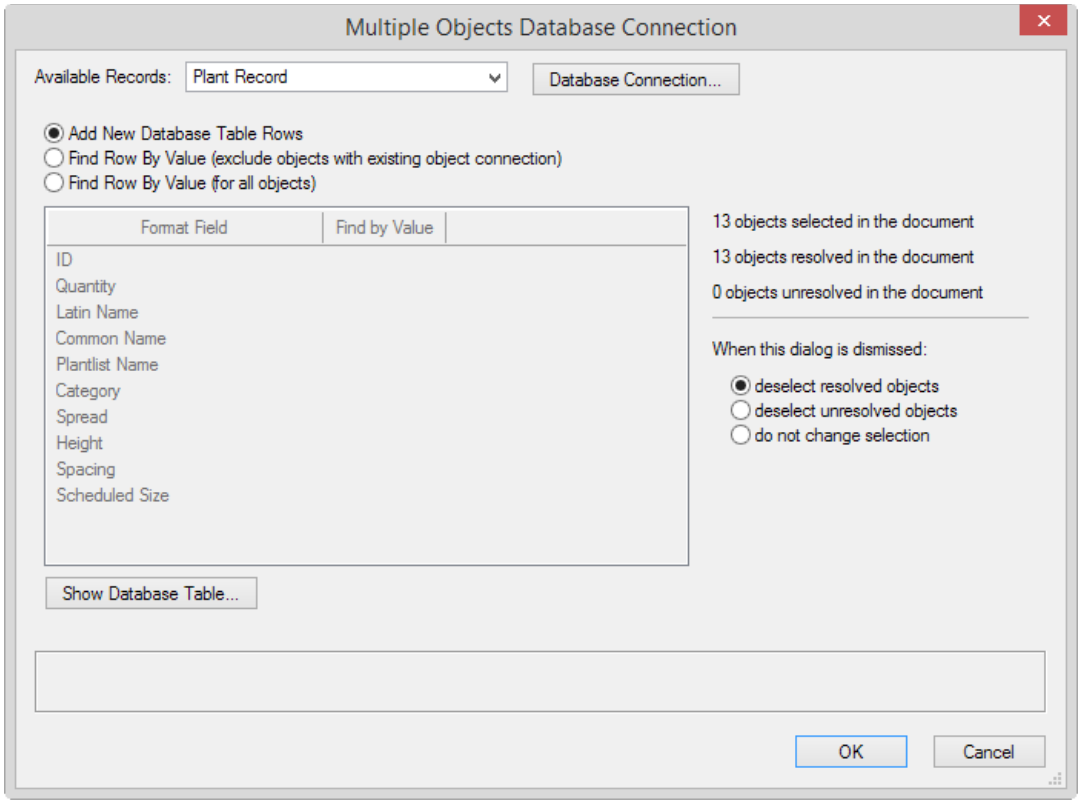
Parameter	Description
Remove	Removes selected table row(s), when one or more record formats does not need to be used as the basis for a database table column. This action does not affect the existing record format in the file.
Selection	Specifies the parameters for the selected table column to be created; the **Database Table Column Name**, **Database Table Column Type**, **Database Link Type**, **Format Field Name,** and **Format Field Type** are suggested based on the record format information, but they can be changed. A column can be created in the database table, but not be connected to a record format field, by selecting <not connected> in the **Format Field Name** list.

3. Specify the parameters of the database table columns to be created. The columns will be created in the order listed; to change the order of a database table column, click and drag within the # column.
4. Click **Close** to return to the Export as Database Table dialog box and make any further necessary table column adjustments.
5. Click **OK**.

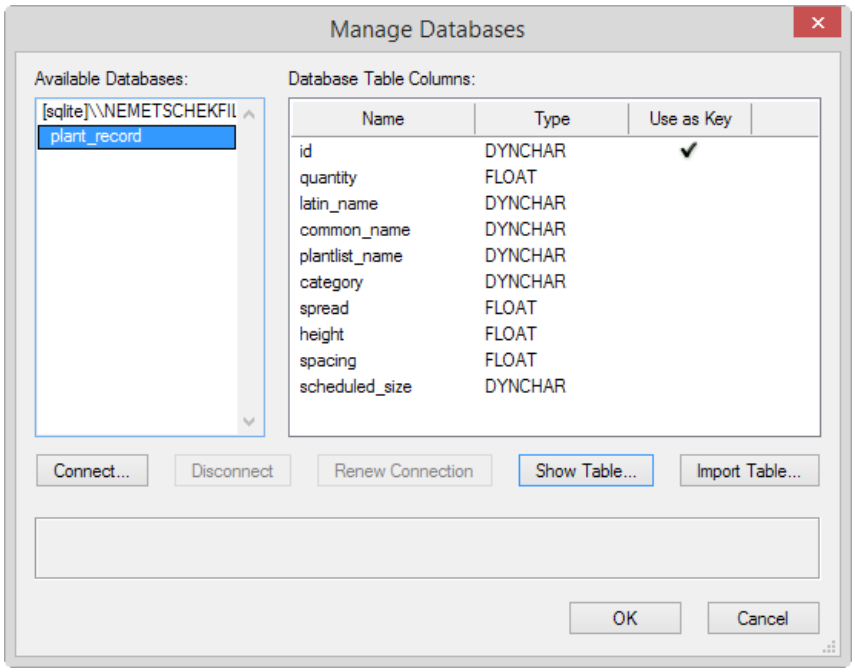
   Each record field in the format becomes a column in the new database table.
6. The database rows are currently empty; the objects in the drawing need to be connected to the table to fill in the data. In this example, the plant record database table was created, so plants in the drawing need to be connected. Select all plants in the drawing.

   Use the **Select Similar** tool to easily select all objects of a particular type that should be connected to the database.
7. Select **Tools > Database > Object Connection**.

   Since several objects were selected, the Multiple Objects Database Connection dialog box opens.

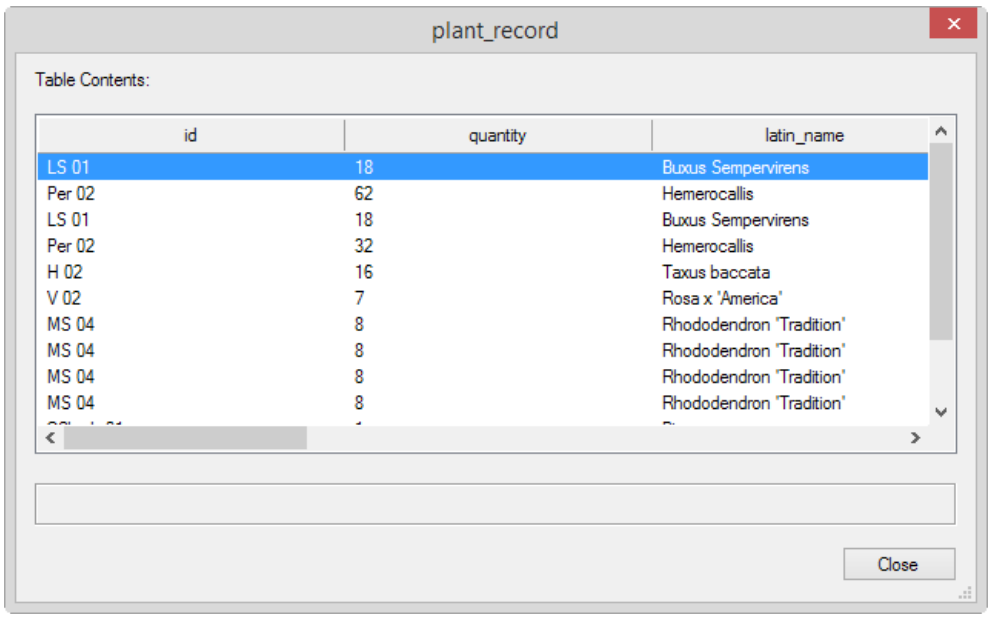


8. Select the applicable record from the **Available Records** list, and then select the **Add New Database Table Rows** option.
9. Click **OK**.
10. To verify that the database rows exist and have been populated with data, select **Tools > Database > Manage Databases**. The database rows have been created for the selected record format.



11. Click **Show Table**.

    A dialog box opens, showing the database table for the associated record format.



12. The objects with the record are connected to the rows of the database table. The table is filled with data from the record formats of the objects in the drawing.

Automatically Creating Record Formats from a Database Table

Specifying Update Settings
Record Format Database Connection

Specifying Update Settings

Setting Update Frequency

Database or Vectorworks file updates can be executed automatically or manually. The update settings control the frequency of automatic updates, or only permit manual updates.

If there are problems connecting to the database, an error message provides troubleshooting information. Click **Details** to open a log file in the default text editor application. The log file provides further information to help diagnose connection errors.

To specify update settings:

1. Select **Tools > Database > Update Settings**.

   The Update Database Settings dialog box opens. Specify the update parameters.

Click to show/hide the parameters.

Parameter	Description
Update database	Sets the frequency of updates from the Vectorworks file to the database
Manually	The database is only updated by selecting the **Update External Database** command
Every record change	Updates the database each time a record field value changes in Record Info
Every __ minutes	Updates the database after a set number of minutes
Update Vectorworks document	Sets the frequency of updates of the Vectorworks file from the database
Manually	The Vectorworks file is only updated by selecting the **Update Vectorworks Document** command
Every __ minutes	Updates the file after a set number of minutes

2. Click **OK**.

Manually Updating the Database

If the record fields have been changed for objects with linked fields to a database table, and the link type for the field is set to **Read/Write** or **Write Only**, the drawing data can update the database table data.

Select **Tools > Database > Update External Database**.

Manually Updating the Vectorworks File

If database information has changed, and the link type for connected record fields is **Read/Write** or **Read Only**, the information can be automatically sent to the Vectorworks file, updating the record fields of all object instances connected to the database.

Select **Tools > Database > Update Vectorworks Document**.

Database Setup
Database Connection

Record Format Database Connection
Object Database Connection
Database-Vectorworks Communication

ODBC Driver Information

When setting up a database program for ODBC functionality, related drivers may not be installed. This list describes drivers available for each platform, where to obtain them, and current problems that may exist.

SQLite database files do not require an ODBC manager.

Windows

On Windows, the ODBC manager is integrated into the operating system (OS).

Driver Name	Driver	Driver Connection	Update Vectorworks records by reading data from database	Update database with Vectorworks record data	Add new database table rows through object connection
Microsoft Access	Integrated* in the OS	Supported, working	Supported, working	Supported, working	Supported, working
Microsoft Excel	Integrated* in the OS	Supported, working	Supported, working	Supported, working	Supported, working
Microsoft SQL Server	If not integrated in the OS, ask vendor	Supported, working	Supported, working	Supported, working	Supported, working
Microsoft Text Driver	Integrated* in the OS	Supported, working	Supported, working	Supported, working	Supported, working
MySQL	Available at: www.mysql.com	Supported, working	Supported, working	Supported, working	Supported, working
FileMaker Pro **	Available at: www.filemaker.com	Supported, working	Supported, working	Supported, working	Supported, working

* On Windows 7 64-bit systems, you need to run the 32-bit ODBC manager (odbcad32.exe) to see the built-in drivers, which are currently 32-bit only.

** FileMaker Pro includes the driver needed to share your database as a data source. To use your FileMaker database file locally, FileMaker Pro must be running, and the database file opened and shared. To share your FileMaker database file as a data source over a network, use FileMaker Server Advanced.

Mac

On the Mac, there is no integrated ODBC manager. Suggestions include:

- ODBC Manager (http://www.odbcmanager.net)
- iODBC Manager (http://www.iodbc.org)
- ODBC Administrator Tool (http://suport.apple.com/kb/DL895)

Driver Name	Driver	Driver Connection	Update Vectorworks records by reading data from database	Update database with Vectorworks record data	Add new database table rows through object connection
Actual SQL	Available at: www.actualtech.com	Supported, working	Supported, working	Supported, working Problem reported: data length restriction when writing to database	Supported, working Problem reported: data length restriction when writing to database
Actual Access	Available at: www.actualtech.com	Supported, working	Not supported	Not supported	Not supported
MySQL	Available at: www.mysql.com	Supported, working	Supported, working	Supported, working	Supported, working
FileMaker Pro *	Available at: www.filemaker.com	Supported, working	Supported, working	Supported, working	Supported, working

* FileMaker Pro includes the driver needed to share your database as a data source. To use your FileMaker database file locally, FileMaker Pro must be running, and the database file opened and shared. To share your FileMaker database file as a data source over a network, use FileMaker Server Advanced.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Database Setup
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Utilities

Obtaining Engineering Properties

The **Engineering Properties** command automatically calculates the engineering properties of a 2D object.

To determine the engineering properties of an object:

1. Select a single object, or select a single object and a locus point.
2. Select **Model > Engineering Properties**.

   The Engineering Properties dialog box opens. The data that displays is selection-dependent.

   For a single closed surface, the following displays:

   - Plane properties (area, perimeter, and absolute coordinates of the centroid of the object)
   - Moments of inertia, section modulus', and radii of gyration about the object's centroidal axes

   For a single closed surface and a locus point, the moments of inertia and radii of gyration about the axes that pass through the locus are also displayed, as well as the horizontal and vertical distances from the locus to the centroid of the object.
3. Select the desired options and units.

Click to show/hide the parameters.

Parameter	Description
Units	Update the displayed information to reflect the selected unit of measurement
Place locus at centroid	Select to add a locus at the centroid of the selected object after closing the Engineering Properties dialog box
Place properties on drawing	Select to place a list of the properties at the next mouse click after closing the Engineering Properties dialog box
Write properties to a file	Select to send the properties to a text file; specify the file name and location after closing the Engineering Properties dialog box

4. Click **OK**.

Obtaining Volumetric Properties

The volumetric properties of a 3D object can be obtained with the **Volumetric Properties** command.

To obtain the volumetric properties of a 3D object:

1. Select the 3D object.
2. Select **Model > Volumetric Properties**.

   The Volumetric Properties dialog box opens, displaying the surface area, volume, and center of mass of the object.

Click to show/hide the parameters.

Parameter	Description
Place locus at center of mass	Places a 3D locus at the center of mass of the object
Place properties on drawing	Places the volumetric properties as text on the drawing at a specified location

3. Set the parameters and click **OK**. If **Place locus at center of mass** was selected, the 3D locus is placed automatically on the object. If **Place properties on drawing** was selected, click in the drawing file to specify the location of the text.

Compressing Images

The bitmap images and image resources in a Vectorworks file can be compressed with the JPEG compression method, to save file space. JPEG compression can significantly reduce bitmap image file size, but can result in the loss of fine detail for some images.

The compression method and file size for a selected image display in the Object Info palette. Images that are already compressed by the JPEG compression method remain unchanged.

A selected bitmap file displays “Bitmap” as the object type at the top of the Object Info palette. A bitmap file may already have had PNG compression applied at import; the **Compress Images** command changes its compression format to JPEG.

Compressing Selected Bitmap Images

To compress selected bitmap images:

1. Select the bitmaps to be compressed.
2. Select **Tools > Compress Images**.

   The Compress Images dialog box opens.
3. Select **Apply JPEG Compression to Selected Bitmap Objects**. Click **OK** to compress the selected images.

Compressing All Bitmap Images and/or Image Resources

The JPEG compression method can be applied to all bitmap images in the file. For the best possible reduction in file size, images that have been imported as resources (shown as image resources in the Resource Browser) can also be compressed by the JPEG compression method.

To compress all bitmap images and/or image resources:

1. Select **Tools > Compress Images**.

   The Compress Images dialog box opens.
2. Select **Apply JPEG Compression to All**. Choose whether to apply the JPEG compression to all bitmap images in the drawing, image resources, or both. Click **OK** to compress the images.

Tracing Bitmaps

The **Trace Bitmap** command traces bitmap objects and picture objects (images which have been imported with the **Import PICT as Bitmap** command). It creates a group of vector lines from the image.

To trace a bitmap or picture object:

1. Select the image to trace.
2. Select **Modify > Trace Bitmap**.

   The Trace Bitmap dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Maximum width in pixels	Specify the maximum number of pixels to be recognized as a single line; pixel counts above the selected value are traced
Collinearity Sensitivity	Select a value to define the segment size and accuracy of the trace

3. Set the parameters and click **OK**.

The time it takes to trace the image can vary from seconds to hours. The tracing time required is determined by the image size, as well as the line threshold and collinearity sensitivity settings selected.

Creating 3D Objects from 2D Objects

The **Create 3D Object from 2D** command places a 3D version of a 2D object in a drawing. The command applies to the following 2D objects with 3D counterparts.

Acorn nut (Inch)	Lock washer (Inch, Metric, DIN, ISO)	Spur gear *
Angle (AISC Inch and Metric, BSI, JIS, ANZ, DIN)	Needle bearing	Spur gear rack
Ball bearing	Nut (Inch, Metric, DIN, ISO)	Square tubing (AISC Inch and Metric, BSI, JIS, ANZ, DIN)
Bearing lock nut	Parallel pin (DIN)	Swing bolt
Bevel gears	Pillow block bearing	Swing eye bolt
Bulb flat (BSI, JIS, DIN)	Plain washer (Inch, Metric, DIN, ISO)	Taper pin (Inch, DIN)
Carriage bolt (Inch, Metric)	Pulley *	Tapered roller bearing
Channel (AISC Inch and Metric, JIS, ANZ, DIN)	Rectangular tubing (AISC Inch and Metric, BSI, JIS, ANZ, DIN)	T-bolt
Clevis pin (Inch, Metric, DIN, ISO	Retaining ring (Inch, DIN)	Tee (AISC Inch and Metric, BSI, JIS, DIN)
Compression spring - 1 and 2	Retaining washer (DIN)	Thrust bearing
Conical compression spring	Rivet - large (Inch)	Thumb screw (Inch)
Cotter pin (Inch)	Rivet - small (Inch)	Torsion spring - Front, End
Die spring	Rivet (DIN)	Tubular rivet (DIN)
Dowel pin (Inch)	Roller bearing	U-bolt
Extension spring - Front, End	Roller chain - circular	Wide flange (AISC Inch and Metric, BSI, JIS, ANZ, DIN)
Eye bolt	Roller chain - linear	Wing nut (DIN)
Flanged bearing - 2 and 4 hole	Roller chain - offset link	Wing nut type A, B, C, D (Inch)
Hole - drilled	Round tubing (AISC Inch and Metric, BSI, JIS, ANZ, DIN)	Woodruff key
Hole - tapped (Inch, Metric)	Screw and nut (Inch, Metric, DIN, ISO)	Wood screw
I-Beam (AISC Inch and Metric, BSI, JIS, ANZ, DIN)	Set screw (Inch, Metric, DIN, ISO)	Worm
J-Bolt (Inch, Metric)	Shaft	Worm gear *
Key	Sheet metal screw (Inch, Metric)	Z-Section
Knurled thumb nut (Inch, DIN)	Shoulder screw (Inch, Metric, DIN, ISO)	
Lag screw (Inch, Metric)	Sprocket *	

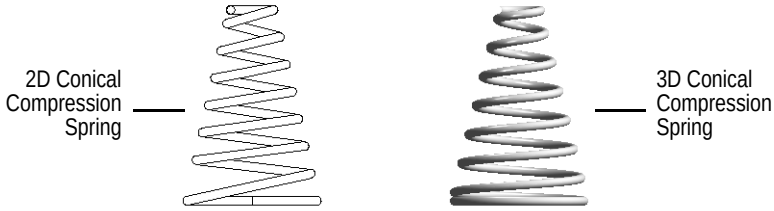
* The spur gear, worm gear, sprocket, and pulley convert to a 3D object and 3D hub object.

This command creates the 3D equivalent of a selected 2D object. If a 2D object with no 3D equivalent is selected, a beep sounds, a notice indicates that the object cannot be converted, and the object is deselected.

To create a 3D object from a 2D object:

1. Select the 2D object. Several 2D objects can be selected at one time.
2. Select the **Create 3D Object from 2D** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Create 3D Object from 2D**
   - Landmark workspace: **Landmark > Machine Design > Create 3D Object from 2D**
   - Spotlight workspace: **Spotlight > Machine Design > Create 3D Object from 2D**

   The 3D object is created with the same parameters as the 2D object.



Spring Calculator

The **Spring Calculator** command solves for spring rates and unit stresses based on compression spring parameters.

To calculate a spring rate:

1. Select the **Spring Calculator** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Spring Calculator**
   - Landmark workspace: **Landmark > Machine Design > Spring Calculator**
   - Spotlight workspace: **Spotlight > Machine Design > Spring Calculator**

   The Spring Calculator dialog box opens.
2. Edit the compression spring parameters. To add to the list of available parameter values, see “Adding User-defined Information to Commands” on page 1883.

Click to show/hide the parameters.

Parameter	Description
Units	Sets the spring dimension units to Inches, cm, or mm
Method	Select the method; as data is entered, the other dimensions are automatically calculated
Dimensions	The fields available depend upon the **Method** and the **Type of Ends** selected
Outside Diameter	Enter the outside diameter
Mean Diameter	Enter the mean diameter
Wire Diameter	Enter the wire diameter
Solid Height	Enter the solid height
No. of Active Coils	Enter the number of active coils

Parameter	Description
Type of Ends	Select **Closed and Ground**, **Open and Ground**, **Closed**, or **Open** to set the type of ends on the spring
Properties	The properties of the spring can be selected from the list or manually entered
Material	Select the material to be used for the spring or select **<Other>** to set the Modulus of Elasticity manually
Mod. of Elasticity	Automatically entered based on the material selected; displayed in pounds per square inch, newtons per square centimeter, or newtons per square millimeter, depending on selected units. If **<Other>** was selected, enter the Modulus of Elasticity manually.
Spring Rate	Automatically calculated based on the spring parameters; displayed in the selected units
Unit Stress	Automatically calculated based on the spring parameters; displayed in the selected units
Place 2D spring on drawing	Select to draw a 2D spring according to the above parameters; if both check boxes are selected, the 2D and 3D springs are automatically aligned in the drawing
Place 3D spring on drawing	Select to draw a 3D spring according to the above parameters; if both check boxes are selected, the 2D and 3D springs are automatically aligned in the drawing

3. Click **Close** to exit the calculator.

Belt Length Calculator

The Belt Length Calculator solves for either belt length or center distance between two pulleys.

To calculate belt length or center distance:

1. Select the **Belt Length Calculator** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Belt Length Calculator**
   - Landmark workspace: **Landmark > Machine Design > Belt Length Calculator**
   - Spotlight workspace: **Spotlight > Machine Design > Belt Length Calculator**

   The Belt Length Calculator dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Find	Select whether to calculate the **Belt Length** or the **Center Distance** between pulleys
(1) Diameter	Enter the first pitch diameter
(2) Diameter	Enter the second pitch diameter
(3) Center Distance	If calculating the belt length, enter the **Center Distance**
Belt Length	If calculating the center distance, enter the **Belt Length**
Place data on drawing	Select to insert the calculated data on the drawing

2. Enter the known values, and then click **Solve**.

   The belt length or center distance value displays.

   If the center distance value is unknown, leave the field blank, and then click **Solve**. The minimum distance is displayed. Click **Solve** again to solve for the belt length based on the minimum center distance.

3. Click **Close** to exit the calculator.

Chain Length Calculator

Chain Length Calculator

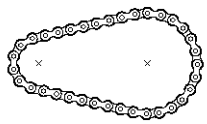
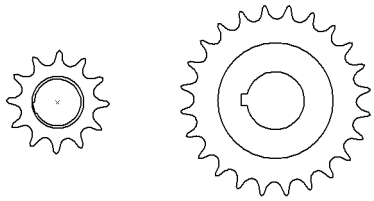
The Chain Length calculator solves for either the length of a chain or center distance between two sprockets.

To calculate chain length or center distance:

1. Select the **Chain Length Calculator** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Chain Length Calculator**
   - Landmark workspace: **Landmark > Machine Design > Chain Length Calculator**
   - Spotlight workspace: **Spotlight > Machine Design > Chain Length Calculator**

   The Chain Length Calculator dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Chain and Sprocket Data	
Find	Select whether to calculate the **Chain Length** or the **Center Distance** between sprockets
Std.Chain No. (Pitch)	Select the pitch of the chain
(1) Number of Teeth, Sprocket #1	Enter the number of teeth of sprocket #1
(2) Number of Teeth, Sprocket #2	Enter the number of teeth of sprocket #2
(3) Center Distance or Chain Length	If calculating the chain length, enter the **Center Distance**; if calculating the center distance, enter the **Chain Length**
Chain Length or Center Distance	Depending on the **Find** selected, the chain length or center distance calculation is displayed
Placement Options	
Place chain on drawing	Inserts the continuous chain onto the drawing, based on the specified and calculated values 
Place sprockets on drawing	Inserts the two sprockets onto the drawing based on the specified and calculated value  Sprocket #1 (N1) Sprocket #2 (N2)

Parameter	Description
Group the objects	Select to group the objects when inserted onto the drawing; if deselected, the objects are ungrouped when inserted onto the drawing and are editable in the Object Info palette
Place data on drawing	When selected, inserts the calculated data onto the drawing

The **Chain Length** value can be entered based on the number of pitches multiplied by the pitch value.

2. Enter the known values, and then click **Solve**.

   The chain length or center distance value displays.

   If the center distance value is unknown, leave the field blank, and then click **Solve**. The minimum distance is displayed. Click **Solve** again to solve for the chain length based on the minimum center distance.

3. Select the desired placement options.
4. Click **OK**.
5. If placement options were selected, the cursor changes to a bull's eye. Click in the drawing to insert the chain and/or sprockets. If **Place data on the drawing** was selected, click again to insert the calculated data.
6. Click **OK** to close the calculator.

Belt Length Calculator

D Control Values for Keys

The Control Values for Keys calculator solves for the key depths of a given shaft and the key size.

To calculate the control values:

1. Select the **Control Values for Keys** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Control Values for Keys**
   - Landmark workspace: **Landmark > Machine Design > Control Values for Keys**
   - Spotlight workspace: **Spotlight > Machine Design > Control Values for Keys**

   The Depth Control Values for Keys dialog box opens.

2. Enter the shaft diameter size. Select **Recommended Key Size** to use the recommended key size according to the ASME or ISO standard; otherwise, select **Custom Key Size** to enter custom key sizes.

Click to show/hide the parameters.

Parameter	Description
Nom. Shaft Diameter (D)	Enter the size of the shaft
Recommended Key Size	Select to use the recommended key size according to the ASME or ISO standard
Square	Select to use the recommended square key
Rectangular	Select to use the recommended rectangular key
Key Width (W)/Height (H)	Displays the dimensions of the recommended key
Custom Key Size	Select to use custom key sizes
Key Width (W)	Enter the width of the key
Key Height (H)	Enter the height of the key

Parameter	Description
Solution	
Depth to Bottom of Key (S)	Displays the dimension from the bottom of the shaft to the bottom of the key
Depth to Top of Key (T)	Displays the dimension from the bottom of the shaft to the top of the key
Place data on drawing	Select to insert the calculated data on the drawing

3. Click **Solve**.

   The key depth values for the given shaft diameter and key size are displayed.

4. Click **Close** to exit the calculator.

D Shaft Analysis

The Shaft Analysis utility analyzes the results of a twisting moment being applied to a round solid or hollow shaft.

To perform the analysis:

1. Select the **Shaft Analysis** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Shaft Analysis**
   - Landmark workspace: **Landmark > Machine Design > Shaft Analysis**
   - Spotlight workspace: **Spotlight > Machine Design > Shaft Analysis**

   The Shaft Analysis dialog box opens.

2. Enter the shaft properties and the known value in the Solutions section. To add to the list of available units, see “Adding User-defined Information to Commands” on page 1883.

Click to show/hide the parameters.

Parameter	Description
Shaft Properties	Enter the shaft properties; the analysis results are based on the properties provided
(1) Outside Diameter	Enter the outside diameter of the shaft; select the desired units from the list
Inside Diameter	Enter the inside diameter of the shaft; enter zero for a solid shaft
Polar Moment of Inertia	Automatically calculated from the outside and inside diameters
(2) Length	Enter the length of the shaft
Shear Modulus	Enter the value of the shear modulus; select the desired units from the list
Solutions	Select and enter the known value
(3) Twisting Moment	Enter the twisting moment; select the desired units from the list
Maximum Shear Stress	Enter the maximum shear stress; the units are set by the Shear Modulus value
(4) Angle of Twist	Enter the angle of twist in degrees

3. Click **Solve**.

   The unknown values in the Solutions section are solved based on the information given.

4. Click **Close** to exit the shaft analysis calculator.

Centroid

The Centroid utility calculates the centroid, or center of gravity, of a 2D shape. The utility shows the location of the centroid and can place a locus at that point. For more information, see “Obtaining Engineering Properties” on page 1821.

To place a centroid locus point on an object:

1. Select the object.
2. Select the **Centroid** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Centroid**
   - Landmark workspace: **Landmark > Machine Design > Centroid**
   - Spotlight workspace: **Spotlight > Machine Design > Centroid**

   The Centroid dialog box opens.
3. The location of the centroid is displayed. Select **Place locus at centroid** to place a locus marker at the centroid of the object.
4. Click **OK**.

   If the object is moved, the locus point does not remain centroidal unless the object and locus point are grouped and then moved.

Conversion Factors

The Conversion Factors utility provides the conversion factor between units.

To perform a conversion factor calculation:

1. Select the **Conversion Factors** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Conversion Factors**
   - Landmark workspace: **Landmark > Machine Design > Conversion Factors**
   - Spotlight workspace: **Spotlight > Machine Design > Conversion Factors**

   The Conversion Factors dialog box opens.
2. In the **Multiply** field, enter the number of units to convert. Select the original unit of measure from the **Multiply** list.
3. Select the target unit of measure from the **To Obtain** list. The conversion results display in the **To Obtain** field and the conversion factor displays in the **By** field.
4. Click **OK** to exit the utility.

Solution of Triangles

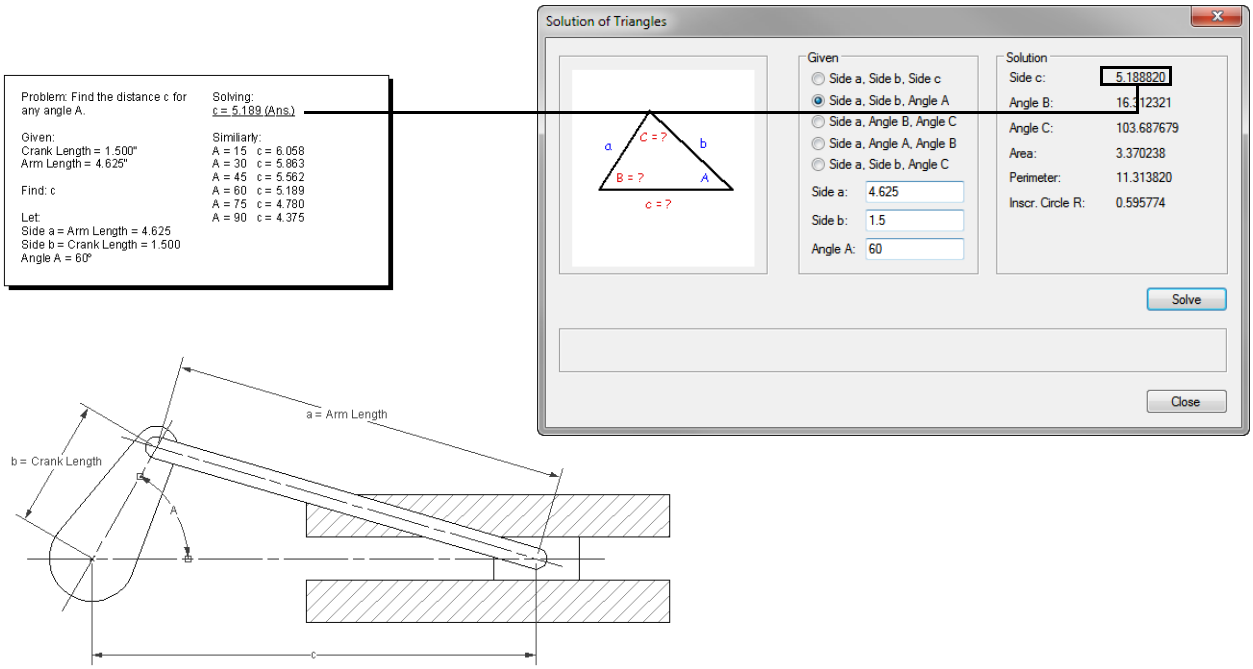
The Solution of Triangles utility solves for unknown values of a triangle.

To solve for the unknown values of a triangle:

1. Select the **Solution of Triangles** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Solution of Triangles**
   - Landmark workspace: **Landmark > Machine Design > Solution of Triangles**
   - Spotlight workspace: **Spotlight > Machine Design > Solution of Triangles**

   The Solution of Triangles dialog box opens.

2. Select the format of the known values, and then enter them in the fields below.
3. Click **Solve**. The calculated values display in the **Solution** fields.



4. Click **Close** to exit the utility.

D 3D Properties

The 3D Properties utility calculates a 3D object's center of mass, radii of gyration, and mass properties based on density or specific gravity, surface area, and volume. This can be used for 3D objects such as a sweep, extrude, or solid.

To display the 3D properties of an applicable object:

1. Select the 3D object.
2. Select the **3D Properties** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > 3D Properties**
   - Landmark workspace: **Landmark > Machine Design > 3D Properties**
   - Spotlight workspace: **Spotlight > Machine Design > 3D Properties**

   The 3D Properties dialog box displays the object surface area, volume, radii of gyration, and center of mass.

Click to show/hide the parameters.

Parameter	Description
Units	Select the unit for displaying values
Places locus at centroid	Select to place a locus point at the 3D center of the object
Place properties on drawing	Select to place a grouped text block containing the 3D property values on the drawing at the next mouse click
Write properties to a file	Select to create a text file containing the 3D property values

3. Click **Calculate Mass Properties**.

   The Mass Properties dialog box displays the weight, mass, and mass moments of inertia of the object.

4. Specify the system of units to use when calculating the mass properties.

   The mass properties calculations display.

Click to show/hide the parameters.

Parameter	Description
Input	
System of Units	Select the units to use for the calculation
Property to Use	Select whether to display properties based on the density or specific gravity
Value	Enter the material density or gravity value

5. Click **OK** to return to the 3D properties dialog box.

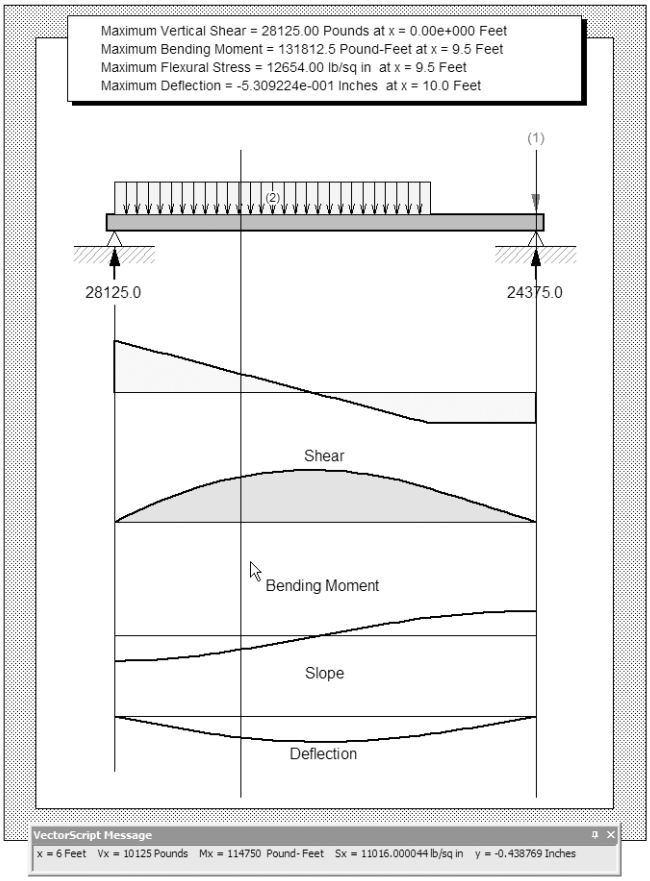
D Simple Beam Analysis

The **Simple Beam Analysis** command opens a message box displaying the calculated values that correspond to the cursor position.

To analyze a simple beam:

1. Create the beam and diagrams as described in “Simple Beam” on page 1888.
2. Select the **Simple Beam Analysis** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Simple Beam Analysis**
   - Landmark workspace: **Landmark > Machine Design > Simple Beam Analysis**
   - Spotlight workspace: **Spotlight > Machine Design > Simple Beam Analysis**

   A message dialog box opens at the bottom of the screen, displaying x (location on the beam), vx (shear), mx (bending moment), sx (shearing stress), and y (deflection) values.



The values displayed depend on the location of the cursor along the beam and the **Calculation Interval** specified in the Beam Properties dialog box.

3. Click on a blank area of the drawing to stop the analysis. Close the message dialog box.

   To lock the values in the message dialog box, click on a point along the beam. The values at this point can then be studied or written down for future analysis. Select the **Simple Beam Analysis** command again to continue checking values along the beam.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Simple Beam
Simple Beam Calculator

D Simple Beam Calculator

The **Simple Beam Calculator** command provides a quick way to analyze a simply-supported beam with a single load.

To use the Simple Beam Calculator:

1. Select the **Simple Beam Calculator** command from the appropriate menu:
   - Architect workspace: **AEC > Framing > Simple Beam Calculator**
   - Landmark workspace: **Landmark > Architectural > Simple Beam Calculator**
   - Spotlight workspace: **Spotlight > Architectural > Simple Beam Calculator**

   The Simple Beam Calculator dialog box opens.
2. Select the desired configuration, and then enter the values to be calculated.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Configuration	Select the simple beam configuration to analyze; the preview window updates to correspond to the selected configuration
Input	
Units	Select Imperial or Metric (SI) as the displayed unit value
Beam and Load Data	Select Imperial or Metric (SI) as the displayed unit value
Beam Length (L)	Enter the length of the beam
Concentrated Load (P)	Enter the value of a concentrated load or loads
Dist. from end of beam (a)	Enter the distance from the end of the beam to the load
Distributed Load (w)	Enter the value per unit length of a distributed load
Beam Properties	
Modulus of Elasticity	Enter the modulus of elasticity of the beam
Moment of Inertia	Enter the moment of inertia of the beam
Section Modulus	Enter the section modulus of the beam
Select Structural Shape and Materials	Opens the Select Structural Shape and Material dialog box for selection of a structural shape, series, size, and material for the beam and axis about which the properties are needed (see “Structural Shapes and Details” on page 443). Click **OK** to return to the Simple Beam Calculator dialog box; the beam properties section is updated with these values for the selected structural shape, and the fields can be manually adjusted as needed.
Solution	
Reaction at Left End (rL)	Displays the reaction at the left support
Reaction at Right End (rR)	Displays the reaction at the right support
Moment at Left End (mL)	Displays the bending moment at the left support
Moment at Right End (mR)	Displays the bending moment at the right support
Max. Shear (vMax)	Displays the maximum vertical shear
Max. Bending Moment (mMax)	Displays the maximum bending moment
Max. Shear Stress	Displays the maximum shear stress
Max. Deflection	Displays the maximum deflection
at x (from left end)	Displays the point from the left end of the beam at which the maximum deflection occurs

Certain input fields may appear dimmed depending on the configuration selected.

3. Click **Solve**.

   The results are displayed in **Solution**.

4. Click **Close** to exit the Simple Beam Calculator dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Simple Beam
Simple Beam Analysis
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Managing Workspaces

The Fundamentals workspace is installed with the Vectorworks Fundamentals product in the folder [Vectorworks]\Workspaces. If the Renderworks product is installed, all of the user interface elements are added to the Fundamentals workspace. Each Vectorworks Design Series product is installed with a unique, industry-specific workspace. Any changes you make to the default workspaces are saved in your User Data and Preferences folder, to maintain the integrity of both the default and custom workspaces. Custom workspaces can also be saved to a workgroup folder (see “User Folders Preferences” on page 57).

To create a custom workspace, you can edit the current workspace, edit a copy of the current workspace, or create a new workspace. Create multiple workspaces for different drawing needs, or customize a single workspace to your personal preferences.

Custom workspaces imported from other users may contain tools or commands that are not available with your current Vectorworks license; if this occurs the Workspace Editor dialog box notifies you which commands and tools in the custom workspace are unavailable for your use. Selecting or attempting to use a tool or command that is unavailable with your license results in a warning notifying you that you cannot use the selected tool or command.

Workspace customization options include:

- Add menus, tool palettes, tool sets, tools, and commands
- Remove unused menus, tool palettes, tool sets, tools, and commands
- Rearrange the order and the position of menus, tool sets, tools, and commands
- Add, modify, delete, and print the keyboard shortcuts for tools and commands
- Configure the context menus
- Establish palette positions and settings to be used in the workspace
- Specify tool set icons

You can also create customized plug-in tools, commands, and objects. See “Using Scripts” on page 1771.

> Between Vectorworks versions, commands may have been added to or removed from a workspace menu; therefore, moving a prior custom workspace to a newer version may result in unintended commands being selected.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating or Editing Workspaces

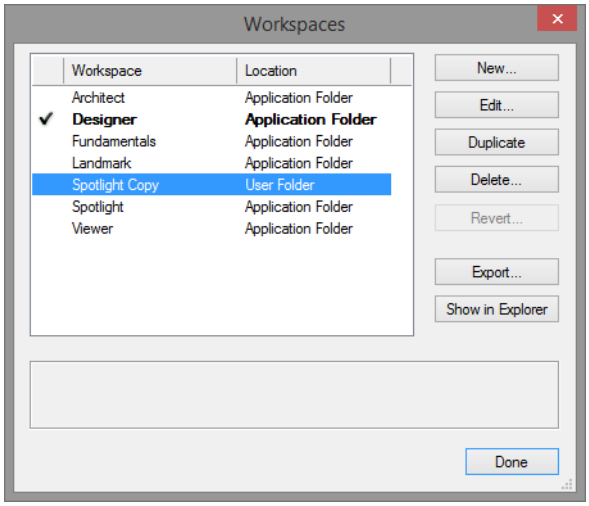
Workspaces can be managed in several ways:

- Select **Tools > Workspaces > Workspaces** to manage all available workspaces
- Select **Tools > Workspaces > Edit Current Workspace** to edit the current workspace
- Select **Customize** from a tool palette’s/ set’s utility menu to edit the active tool palette/set portion of the workspace

To manage all workspaces:

1. Select **Tools > Workspaces > Workspaces.**

   The Workspaces dialog box opens.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Option	Description
New	Opens the Workspace Editor dialog box and creates a new workspace
Edit	Opens the selected workspace in the Workspace Editor dialog box. Only one workspace can be edited at a time. If you edit one of the standard Vectorworks workspaces, Vectorworks will save the customized copy in the user folder and hide the original in the application folder.
Duplicate	Creates a duplicate of the currently selected workspace. Copies automatically have “copy” appended to the end of the original workspace name.
Delete	Deletes the selected workspace permanently from the User folder. Default workspaces cannot be deleted from the application folder. If you delete the active workspace, Vectorworks will re-load with the first workspace in the list as the active workspace.
Revert	Deletes the selected workspace from the user folder if a workspace with the same name exists in the Workgroup or application folder. You cannot revert a workspace in the Workgroup folder. If used on a customized workspace from the application folder, reverts the workspace to its default settings. If you revert the active workspace, Vectorworks will reload the workspace with the same name from the application folder.
Export	Exports the selected workspace as a text file. To export a list of all menu items, tools, and keyboard shortcuts contained in the current workspace, click **Export**. When prompted, specify the file name and the location for the file and click **OK**. The text file also lists keys reserved by the Vectorworks program or the operating system, shortcuts that cannot be user modified, and shortcuts that can be modified from within the Workspace Editor.
Reveal in Finder (Mac) Show in Explorer (Windows)	Opens the folder containing the selected workspace

2. Select a workspace to edit from the list and click **Edit**, or click **New** to create a new workspace.

   The Workspace Editor dialog box opens.

Workspaces in different folder locations with identical names will not be listed, Workgroup folder workspaces override Application folder workspaces, and User folder workspaces override both Workgroup and Application folder workspaces.

3. Modify the menus, context menus, tools, keyboard shortcuts, and palette positions and settings as described in the following sections:
   - “Modifying Menus and Commands” on page 1837
   - “Modifying Context Menus” on page 1839
   - “Modifying Tool Palettes and Tool Sets” on page 1840
   - “Modifying Snapping and Mode Shortcuts” on page 1842
   - “Saving Initial Palette Positions and Settings” on page 1843

   Some shortcuts are reserved by the Vectorworks program or by the Windows or Mac operating system. The Workspace Editor does not prevent such shortcuts from being assigned to a palette or a tool. If this occurs, the Windows or Mac shortcut typically overrides the shortcut assigned in the Vectorworks program.

   If a duplicate item is added to a menu, a palette, or a tool set, the item displays in both locations. Highlight the undesired occurrence of the tool or command and press the Delete key to remove it.

4. When you are finished customizing or creating workspaces, click **Done** to close the Workspace dialog box.

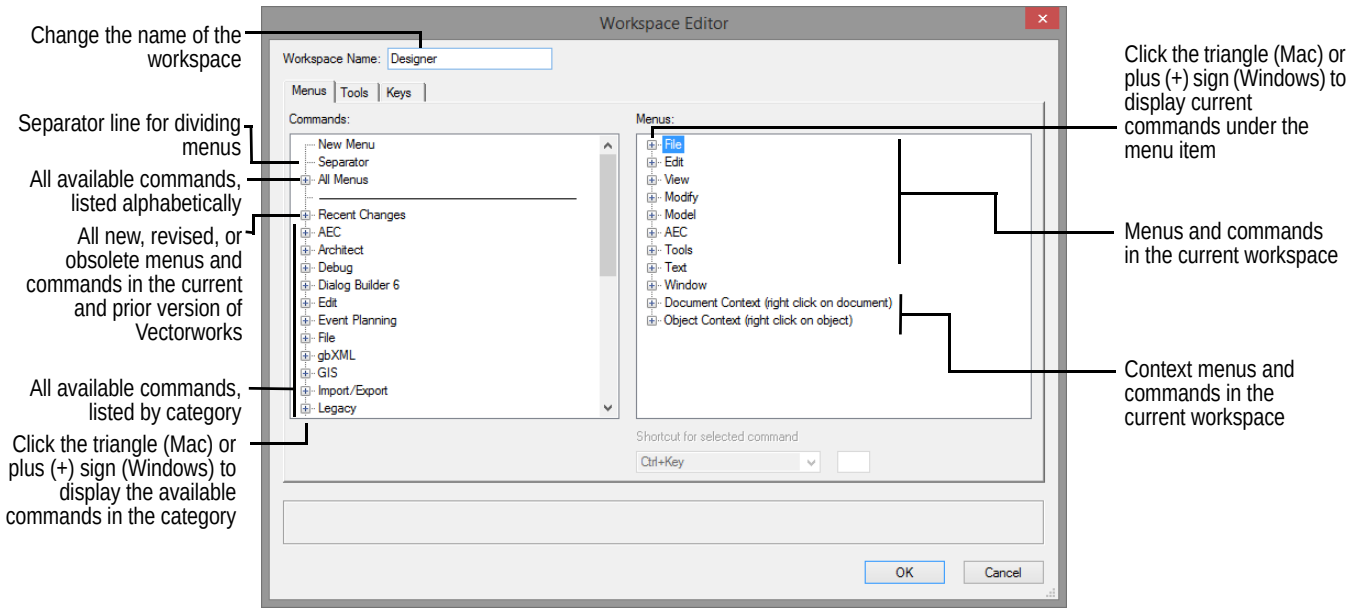
Modifying Menus and Commands

Modify menus and assign keyboard shortcuts to commands, as needed.

To add, modify, or delete a menu or a command:

1. Access the Workspace Editor dialog box as described in “Creating or Editing Workspaces” on page 1835.

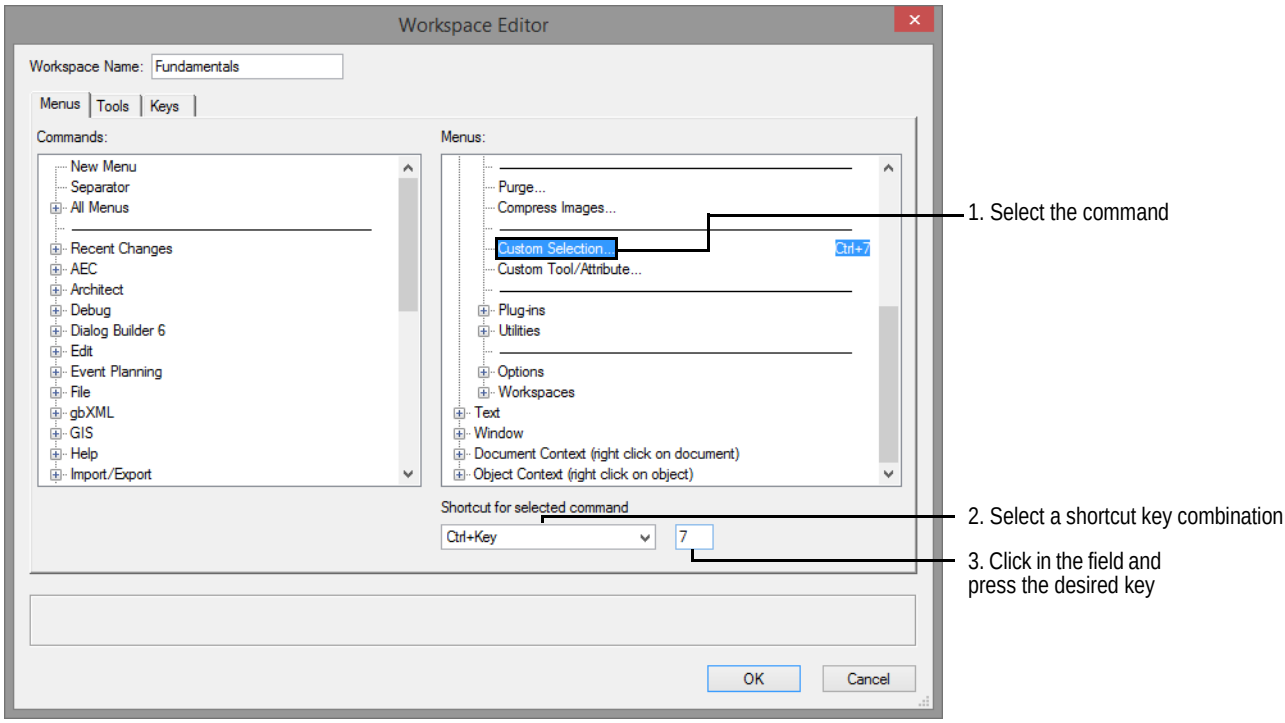
   Click the Menus tab. The left side of the dialog box displays all available commands, grouped by category. The right side of the dialog box displays the menus and commands currently assigned to the workspace. To help you easily identify what changed in this release, the Recent Changes folder lists the new and revised menus and commands for the current and prior version of the Vectorworks software; it also lists those menus and commands that are now considered obsolete (legacy). Modify the menus and commands as described in the following table.



Option	Description
Add a new menu or submenu	Click the **New Menu** command from the Commands list and drag it to the desired position in the Menus list. Click to highlight the new item, and then click again to edit it; enter the name of the new menu item.
Add a command to a menu	Click the plus sign (Windows) or triangle (Mac) to expand the list of commands; click-drag the command from the Commands list to the desired position in the Menus list
Add a separator	Click-drag the **Separator** command (Windows) or separator line (Mac) from the Commands list to the desired position in the Menus list; a separator line displays
Move an item	Click-drag the item in the Menus list to the desired position
Delete an item	Select the item from the Menus list and press the Delete key. If you delete an item that has other items stacked beneath it, all of the stacked items are deleted along with the main item; to prevent this, move the stacked items to another location first. A warning will display asking you to confirm deletion of an item if other items are stacked beneath it.
Change a menu name	Select the menu from the Menus list and type the desired name; commands cannot be renamed

The Document Windows, Font, Tool Palettes, and Workspaces menu items are populated at run time. Therefore, they can only be placed as the last item in a list of submenu items.

2. If desired, assign or change a combination of keys to use as a shortcut to access a menu command.



If a keyboard shortcut is already in use, the option to reassign the shortcut to the current menu command is presented. If the shortcut is reassigned, the original command no longer has a shortcut.

Common computer keyboard shortcuts such as Ctrl + C (Windows) or Cmd + Z (Mac) and operating system shortcuts cannot be overridden.

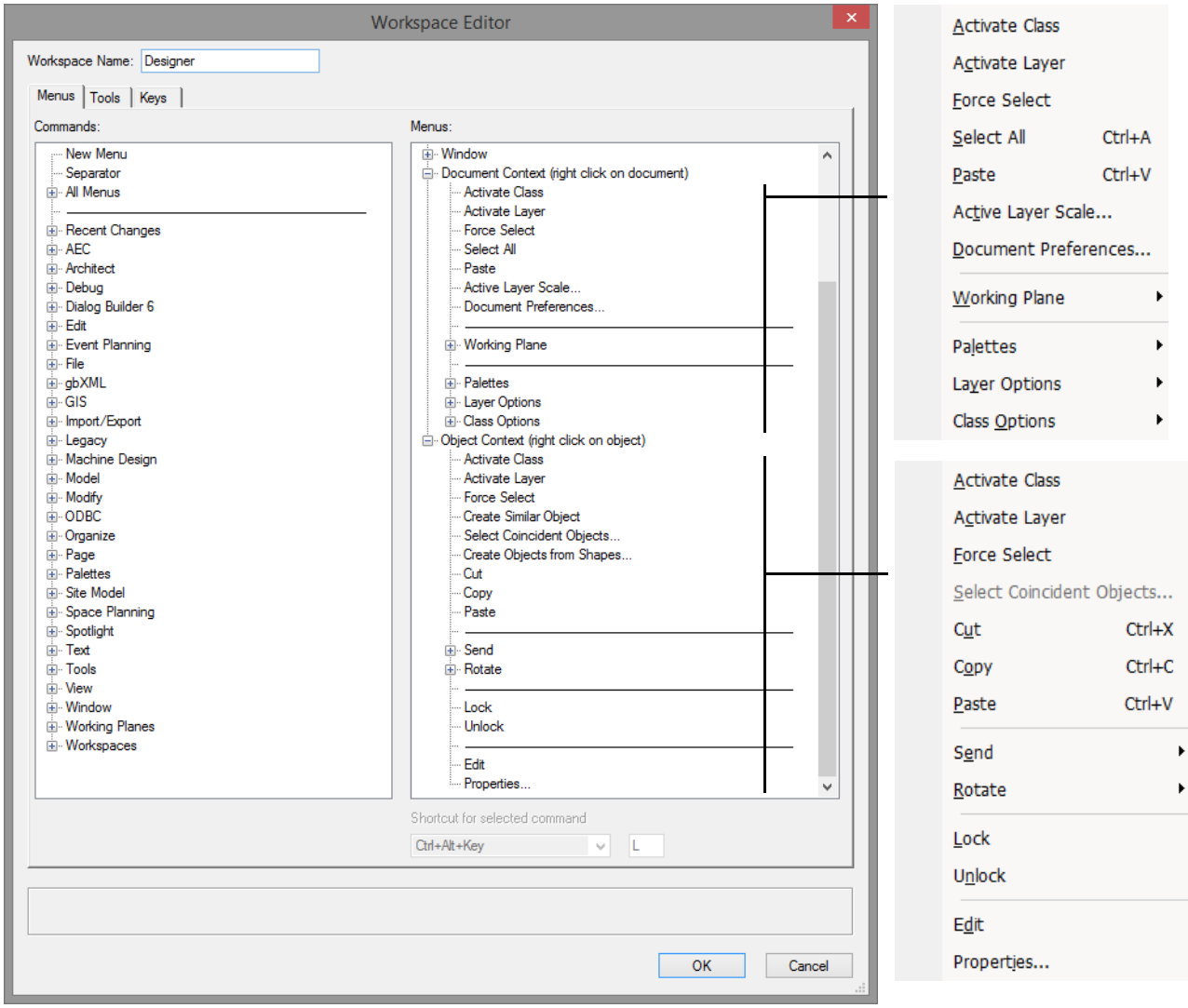
3. If necessary, modify or delete a keyboard shortcut.
   - **Modify**: Select the shortcut from the Menus list; select a new shortcut key combination and/or enter a different key for the shortcut (if that key is reserved by the Vectorworks program or is already in use, a message displays)
   - **Delete**: Select the shortcut key from the Menus list and press Delete
4. Click **OK** to save the changes and close the Workspace Editor dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating or Editing Workspaces
Modifying Context Menus

Modifying Context Menus

Modify document context menus and object context menus in the same manner as described in “Modifying Menus and Commands” on page 1837. These menus display when you click on an object or on the drawing area with a right-click (Windows) or Ctrl-click (Mac). The object context menus contain context-sensitive commands that pertain to the item that is currently selected in the drawing area.


~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating or Editing Workspaces

Modifying Tool Palettes and Tool Sets

Modify tool palettes and tool sets, and assign keyboard shortcuts to tools, as needed. Tools can only be added to tool sets, not to tool palettes.

To provide more drawing space, stack the tools that have similar functionality.

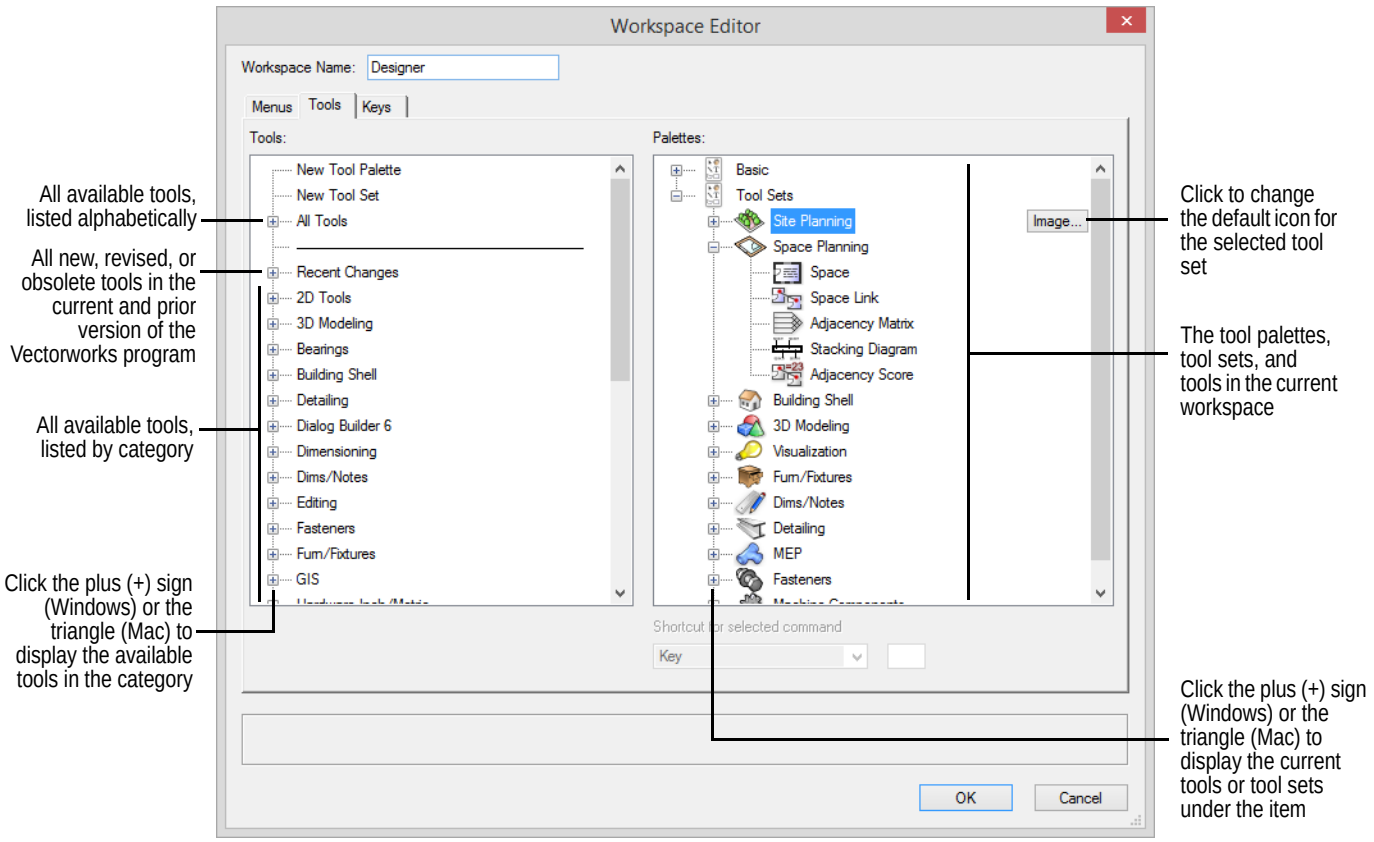
To add, modify, or delete a tool palette or a tool set:

1. Access the Workspace Editor dialog box as described in “Creating or Editing Workspaces” on page 1835.

   Alternatively, you can directly edit a tool set by clicking **Customize** from the tool palette Utility menu.

2. Click the Tools tab. The left side of the dialog box displays all available tools, grouped by category. The right side of the dialog box displays the tool palettes, the tool sets, and the tools that are currently assigned to the workspace. To help you easily identify what changed in this release, the Recent Changes folder lists the new and revised tools for the current and prior version of the Vectorworks software; it also lists those tools that are now considered obsolete (legacy). Modify the tool palettes and the tool sets as described in the following table.

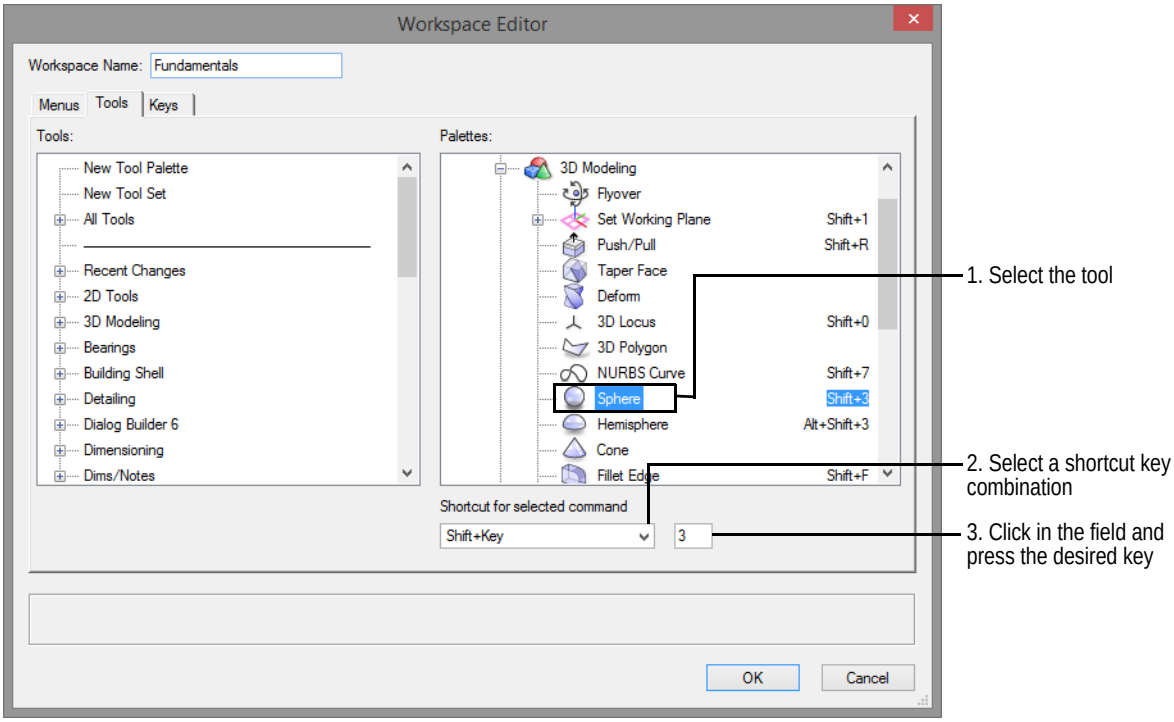
   For the Basic palette, the edit tools and the tools for 2D and 3D object creation are listed in the View/Draw tool set.



Option	Description
Add a new tool palette	Click-drag the **New Tool Palette** command from the Create list to the Palettes list; click the item’s text label and type the new palette’s name in place of **New Tool Palette**

Option	Description
Add a new tool set	Click-drag the **New Tool Set** command from the Create list to the Palettes list; click the item's text label and type the new tool set's name in place of **New Tool Set**
Change a tool set icon	The default icon for the tool set can be replaced by a custom icon, if desired. With a third-party icon editor, create two versions of the icon: one for a standard-resolution display and one for a high-resolution display: • Standard-resolution: an image centered in an area 26 pixels wide by 20 pixels high. Save the icon in .png format. • High-resolution: an image centered in an area 52 pixels wide by 40 pixels high. Save the icon in .png format with the same name as the standard-resolution image with "@2x" appended to the end of the file name. For instance, if the standard-resolution icon is named 3DModelingToolSet.png, the high-resolution icon must be named 3DModelingToolSet@2x.png; otherwise the Vectorworks software will be unable to locate the icon. Select the tool set and click **Image** to import the standard resolution icon; if a high-resolution icon is also present with the @2x designation, the software automatically imports it as well. The Vectorworks program automatically displays the appropriate image for the device's resolution.
Add a tool or a tool category to a tool set	Click the plus sign (Windows) or triangle (Mac) to expand the list of tools; click-drag the tool (or the entire group of tools) from the Tools list to the desired position in the Palettes list
Move an item	Click-drag the item in the Palettes list to the desired position
Delete an item	Select the item in the Palettes list and press the Delete key. If you delete an item that has other items stacked beneath it, all of the stacked items are deleted along with the main item; to prevent this, move the stacked items to another location first.
Change the name of a tool palette or a tool set	Select the item from the Palettes list, click the item's text label, and type the desired name; tools cannot be renamed

3. If desired, assign or change the combination of keys to use as a shortcut to access a tool.



If a keyboard shortcut is already in use, the option to reassign the shortcut to the current tool is provided. If the shortcut is reassigned, the original tool no longer has a shortcut.

Common computer keyboard shortcuts such as Ctrl + C (Windows) or Cmd + Z (Mac) and operating system shortcuts cannot be overridden.

4. If necessary, modify or delete a keyboard shortcut.
   - **Modify**: Select the shortcut from the Palettes list; select a new shortcut key combination and/or enter a different key for the shortcut (if that key is reserved by the Vectorworks program or is already in use, a message displays)
   - **Delete**: Select the shortcut key from the Palettes list and press Delete
5. Click **OK** to save the changes and close the Workspace Editor dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating or Editing Workspaces

Modifying Snapping and Mode Shortcuts

You can modify the keyboard shortcuts for switching between modes on the Tool bar, and for activating snapping functionality.

To modify the shortcuts for modes and snapping:

1. Access the Workspace Editor dialog box as described in “Creating or Editing Workspaces” on page 1835.
2. Click the Keys tab to display the keyboard shortcuts for the workspace currently in use. Assign or modify the keyboard shortcuts, as needed.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Mode Groups	Sets the shortcut keys to switch modes for the active tool; the key in **1** controls the first mode group, the key in **2** controls the second mode group, and so on
Other Keys	Sets the keyboard shortcuts for the functions below
Toggle SmartCursor Cues	Press this key to toggle on and off the SmartCursor cue display
Toggle Angles Relative To Prior Segment	Press this key to toggle whether angle snaps are set relative to the axes or to the previously drawn segment
Set Datum	Press this key to create a datum point for snapping at the current cursor location
Snap Loupe	Press this key to open the Snap Loupe window
Set Smart Point, Edge, or Vector Snap Lock	Press this key to set a smart point, smart edge, or vector snap lock at the current location
Toggle Automatic Working Plane	Press this key to toggle on and off the automatic working plane functionality; see "The Automatic Working Plane" on page 153
Suspend Snapping	Press and hold this key to disable snapping temporarily
X-ray Select Mode	Press and hold this key to see and select objects that are normally obscured behind 2D object fills or 3D rendered objects; see "X-ray Select Mode" on page 113
Coincident Selection	If the cursor is over multiple objects at the current location, press and hold this key and click to open a dialog box from which you can select one or more of the objects; see "Coincident Object Selection" on page 112
Close Polygon, Polyline, Wall Network or Path Objects	After completing all but the final click, press the keyboard shortcut to automatically close polygon, polyline, polygonal wall network, or path-based objects
Snapping Keys	Sets the keyboard shortcuts to toggle on and off each Snapping palette tool

3. Click **OK** to save the changes and close the Workspace Editor dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Vectorworks Workspace
SmartCursor Cues
Angle Snapping
Smart Points Snapping
Smart Edge Snapping
Creating Vector Locks
Using the Snap Loupe
Creating or Editing Workspaces

Saving Initial Palette Positions and Settings

You can save a set of initial palette positions and settings for a workspace. For example, a CAD manager might want to create a customized workspace—including palette settings—to be shared with all new users.

This command simply creates the initial settings for the first time a workspace is used. Each time a user exits the Vectorworks program, the current palette settings and positions are automatically saved for each workspace in a file in the user folder ([User]\Settings\SavedSettings.xml). The settings in this user file override the settings in the workspace file. To reset to the original workspace settings, click **Reset Saved Settings** from the Session tab of Vectorworks preferences (see "Session Preferences" on page 52).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To save initial palette positions and settings for a workspace:

1. Customize the workspace as needed in the Workspace Editor.
2. Place palettes in the desired position, set palette size, and set whether palettes should be minimized, or docked (Windows only); see “Palette Layout Options” on page 40, “Minimizing Palettes” on page 41, and “Docking Windows Palettes” on page 41.
3. For tool palettes, set the tool and tool set display and sort style, and the tool set placement option; see “Tool Palette Features” on page 42.
4. For the Object Info palette, click the Shape, Data, or Render tab to set the active pane upon opening, and manually position the separator in the Data pane, as desired.
5. For the Resource Browser, set the resource display style and group visibilities; see “Viewing Resources” on page 223 and “Hiding and Showing Resources” on page 225.
6. For the Navigation palette (Vectorworks Design Series required), click the desired tab to set the active pane upon opening.
7. Once palettes are positioned and settings have been established, select **Window > Palettes > Save Palette Positions**. A message displays the location of the workspace file where the changes were saved. Click **OK**.
8. Place the custom workspace file in a user Workspaces folder to make it available in the Vectorworks program. When that workspace is selected, the initial palette settings display.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating or Editing Workspaces

Updating Custom Workspaces and Plug-in Objects

When a Vectorworks software upgrade is installed, the workspaces and plug-in objects in the Vectorworks root folder (where the application is installed) are replaced.

Your custom workspaces and plug-ins (in your user folder) remain in place; see “User Folders Preferences” on page 57 for details about user folders. The first time the new version of the Vectorworks program is opened, it automatically converts the custom workspaces to the new format and saves a copy of the original workspaces in a Legacy Workspaces folder (in your user folder).

The Vectorworks installer and updater programs both make backup copies of all workspaces. The installer places backup copies of workspaces in a folder entitled Original, within the [Vectorworks]\Workspaces folder. The updater places backup copies of workspaces in a sequentially numbered folder entitled Backup, within the Workspaces folder. Each time the updater is run, another Backup folder is created. After the backup, both the installer and the updater overwrite the workspaces at the top level of the Workspaces folder.

For more information about where plug-in objects may be located, see “Plug-in File Location” on page 1781.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Standards

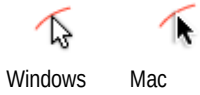
Vectorworks Cursors

The cursor in the Vectorworks program changes depending on which tool, mode, and modifier keys are currently in use, as well as what type of object the cursor is over in the drawing area. This helps you to know which actions are available at any given time, without shifting your attention from the drawing.

Cursor	Description
2D Polygon tool	
	Inner Boundary mode is active; click in an enclosed area to create a 2D polygon based on the area's inner boundary
Reshape tool (2D modes)	
	Move Polygon Handles mode is active, and the cursor is over a handle; click-click to move the handle
	Move Edges Parallel mode is active, and the cursor is over a handle; click-click to move the handle
	Change Vertex mode is active, and the cursor is over a vertex handle; click to change the vertex to the type selected in the Tool bar
	Add Vertex mode is active, and the cursor is over a handle; click-drag the handle to create a new vertex
	Delete Vertex mode is active, and the cursor is over a vertex handle; click to delete the vertex
	Rectangular Marquee mode is active, and the cursor is not over any handle
	Lasso Marquee mode is active, and the cursor is not over any handle
	Polygonal Marquee mode is active, and the cursor is not over any handle
Reshape tool (3D modes)	
	The cursor is over a handle on a selected 3D object that can be dragged vertically (for example, the top handle of an extrude)
	The cursor is over a handle on a selected 3D object that can be dragged in any direction (for example, any handle on a NURBS curve)
	The cursor is over a handle on a selected 3D object that can be scaled (for example, a radius handle on a sphere)
	Change Vertex mode is active, and the cursor is over a vertex handle; click to change the vertex type
	Add Vertex mode is active, and the cursor is over a handle; click-drag the handle to create a new vertex
	Delete Vertex mode is active, and the cursor is over a vertex handle; click to delete the vertex

Cursor	Description
Align Plane tool	
	Click to define the X and Y axes on the selected object
Angular Dimension tool	
	Edges mode is active; click two intersecting objects to create the dimension object
Attribute Mapping tool	
	Click-drag to scale the fill larger or smaller (the cursor is over a corner handle of the mapping object)
	Click-drag to rotate the fill (the cursor is over a middle handle of the mapping object)
 Near a handle  Anywhere except a handle	Click-drag to move the origin of the fill (the cursor is over other parts of the mapping object)
Center Mark tool	
	Click the highlighted object to create a center mark
Chamfer Edge tool	
Select Faces preference off  Windows  Mac	The cursor highlights edges that can be used in the chamfer operation
Select Faces preference on  Windows  Mac	The cursor highlights faces that can be used in the chamfer operation
Constraint tools	
	Click the objects to be constrained
Edit Curtain Wall tool	
	Click to add a new frame to the curtain wall
	Click to split the frames at the frame intersection
	Click on the second collinear frame to combine the frames
	Click to move the frame grid along the curtain wall

Cursor	Description
Extract tool	
Extract Point or Extract Curve mode  Windows Mac	The cursor highlights edges that can be used in the extract operation
Extract Iso-parametric Curve or Extract Surface mode  Windows Mac	The cursor highlights faces that can be used in the extract operation
Eyedropper tool	
	Pick up Attributes mode is active
	Apply Attributes mode is active
Fillet Edge tool	
Select Faces preference off  Windows Mac	The cursor highlights edges that can be used in the fillet operation
Select Faces preference on  Windows Mac	The cursor highlights faces that can be used in the fillet operation
Fixed Point Resize tool	
	Click the point on the selected object that is to remain fixed, and then move the cursor to resize the object
Flyover tool	
	Click-drag to “fly over” the drawing
Help (What’s This? command)	
	Click a tool or command to open the online help topic that describes it

Cursor	Description
Loft Surface tool	
 Windows Mac	The cursor highlights curves that can be used in the loft operation
Move Page tool	
	Click to move the page print boundary
Pan tool	
	The **Pan** tool is active, or the pan mode is activated when another tool is active (Space bar is pressed)
Polyline tool	
	Bézier Vertex mode
	Cubic Vertex mode
	Arc Vertex Fillet mode
Project tool	
Windows Mac	The cursor highlights curves that can be used in the project operation
Push/Pull tool	
Windows Mac	In Extrude Face or Move Face mode, the cursor highlights faces that can be used in the push/pull operation
Radial Dimension tool	
	Click the highlighted circle to create a radial dimension
Render Bitmap tool	
	Click-drag to select the area from which to create a rendered bitmap image
Rotate View tool	
	Click-drag to rotate the drawing view
Selection tool	
	Click to scale the selected object
	The cursor is over an object; click to select or click-drag to move the object

Cursor	Description
	Click-drag to duplicate the object (the Ctrl key is pressed on Windows, or the Option key is pressed on Mac)
	The cursor is over an object; click to select or click-drag to move the object
	Click to add an object to the current selection (the Shift key is pressed)
	Click-drag to duplicate an object (the Ctrl key is pressed on Windows, or the Option key is pressed on Mac)
 Windows  Mac	The cursor is not over any selectable object
	Lasso Marquee mode is active
	Polygonal Marquee mode is active
	The cursor is over edges or points of multiple objects; use either the Coincident Selection key or the **Select Coincident Objects** command to select the appropriate objects
	The cursor is over edges or points of multiple objects, and the Shift key is pressed; use either the Coincident Selection key or the **Select Coincident Objects** command to select the appropriate objects to add to the selection
	The cursor is over edges or points of multiple objects; use either the Coincident Selection key or the **Select Coincident Objects** command to select the appropriate objects to move
	The cursor is over the resizing points of multiple objects; use either the Coincident Selection key or the **Select Coincident Objects** command to select the appropriate objects to resize
Set Working Plane tool	
	Click to place the working plane
	Click the square center grip and move the cursor to move the working plane
	Click a red, green, or blue axis grip and move the cursor to rotate the working plane
Shear tool	
	Click the point on the selected object that is to remain fixed, and then move the cursor to shear the object
Shell Solid tool	
 Windows  Mac	The cursor highlights faces that can be used in the shell solid operation

Cursor	Description
Symbol Insertion tool	
	Symbol Pick up mode is active; click a symbol to make it active
Text tool	
	Click to create a text object
	Click to place the text editing cursor in a text object or in a field
Translate View tool	
	Translates the drawing view from side to side or up and down
	In perspective projection, adjusts the distortion by moving the mouse up (more distorted) or down
Trim tool	
	Click to trim the object under the cursor
Walkthrough tool	
	Walk mode; Click-drag to simulate a walkthrough by moving through the drawing
	Look Around mode; Click-drag to change the viewing angle up, down, left, or right
	Elevator mode; Click-drag to move the viewer higher or lower
	The cross indicates the control point of the walkthrough
Worksheet object	
	The cursor is over a worksheet cell
	The cursor is over a worksheet column header cell
	The cursor is over a worksheet row header cell
	Click-drag left or right to change the column width
	Click-drag up or down to change the row height
	Click-drag left/right to remove/add columns; click-drag up/down to remove/add rows
 Windows  Mac	Click-drag to move the column or row to a new location

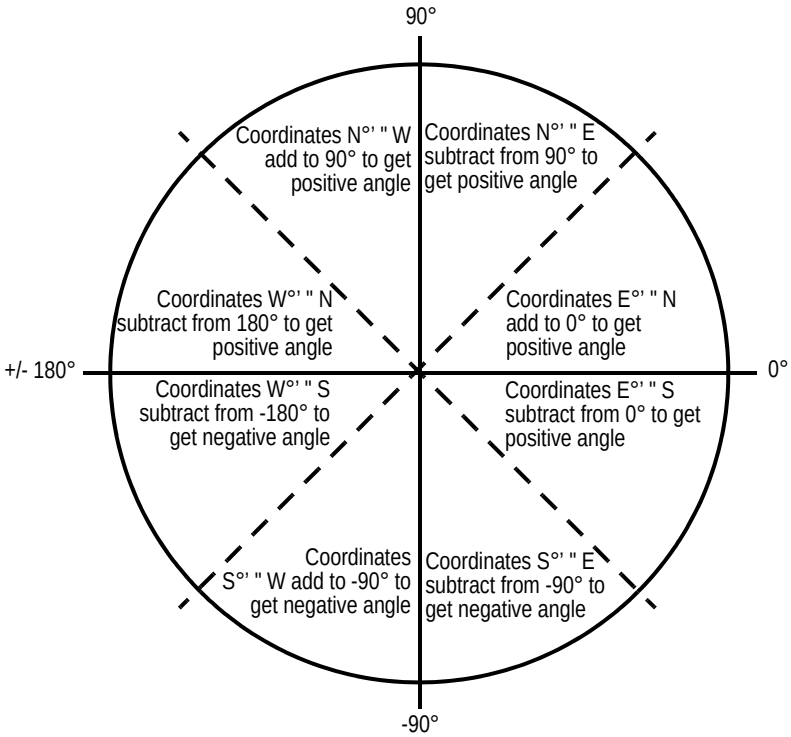
Cursor	Description
Zoom tool	
	Marquee mode is active; click and move the cursor to create a marquee around the area to zoom in on
	Marquee mode is active, and the Alt key (Windows) or Option key (Mac) is pressed; click and move the cursor to create a marquee around the area to zoom out on
	Interactive mode is active; click-drag up to zoom in, or click-drag down to zoom out
Context menus (Mac only)	
	The control key is pressed while the cursor is over an object or an open area in the drawing; click to open the context menu
All remaining tools	
+	Click to begin the tool operation (such as placing a new drawing object, or splitting an existing object)

Vectorworks Circle/Arc Conventions

Due East is 0º. Positive degrees run counter-clockwise and negative degrees run clockwise.

Survey Bearings

The acute angle between the Meridian and a line measured from North to South, toward East and West gives a reading of less than 90º.



Correlated Color Temperature

Color temperature is a simplified way to indicate the hue of a light source in degrees Kelvin (K). Lower color temperatures suggest more of a yellow-reddish color, while higher color temperatures tend to be more blue.

The following chart can be used to approximate various light sources in a rendered scene.

Approximate Degrees K	Light Source Example
1500 - 1800	Candlelight
2800 - 3200	Indoor household tungsten light bulb
5000 - 5500	Sunny day at noon
6500 - 7000	Overcast sky
9000 - 12000	Blue sky

Architectural Scale

The following table provides the architectural scale conversion.

1" = 1' Fraction of inch equaling 1 foot	12 x 1/1 = 12 Inches/ foot multiplied by inverted fraction	1 : 12 Number to be typed into Paper Scale
1/2" = 1'	12 x 2/1 = 24	1 : 24
1/4" = 1'	12 x 4/1 = 48	1 : 48
3/4"= 1'	12 x 4/3 = 16	1 : 16
1/8" = 1'	12 x 8/1 = 96	1 : 96
3/8" = 1'	12 x 8/3 = 32	1 : 32
5/8" = 1'	12 x 8/5 = 19.2	1 : 19.2
7/8" = 1'	12 x 8/7 =13.714...	1 : 13.7142857
1/16" = 1'	12 x 16/1 = 192	1 : 192
3/16" = 1'	12 x 16/3 = 64	1 : 64
5/16" = 1'	12 x 16/5 = 38.4	1 : 38.4
7/16" = 1'	12 x 16/7 = 27.428...	1 : 27.4285714
9/16" = 1'	12 x 16/9 = 21.333...	1 : 21.3333333
11/16" = 1'	12 x 16/11 = 17.454...	1 : 17.4545454
13/16" = 1'	12 x 16/13 = 14.769...	1 : 14.7692307
15/16" = 1'	12 x 16/15 = 12.8	1 : 12.8
1/32" = 1'	12 x 32/1 = 384	1 : 384
3/32" = 1'	12 x 32/3 = 128	1 : 128
5/32" = 1'	12 x 32/5 = 76.8	1 : 76.8
7/32" = 1'	12 x 32/7 = 54.857	1 :54.8571428
1/64" = 1'	12 x 64/1 = 768	1 : 768
3/64" = 1'	12 x 64/3 = 256	1 : 256

Vectorworks Equivalent to AutoCAD Terms and Concepts

Many concepts are similar between Vectorworks and AutoCAD® software; however, different vocabulary can result in confusion for AutoCAD users new to Vectorworks. Additionally, some features of the Vectorworks program have no equivalent in AutoCAD, and vice versa.

Layers in the Vectorworks program are not the same as layers in AutoCAD (instead, Vectorworks classes are the equivalent). Vectorworks layers are like sheets of vellum, placed one on top of the other, with all layers visible and editable, or only the current layer showing/editable. Like hand drafting, each design layer of a Vectorworks drawing has a scale. Sheet layers, which contain viewports, are at a scale of 1:1 and are the "paper space" equivalent.

In the Vectorworks program, some tool is always active; switch tools by selecting a different tool.

The help system includes a PDF listing all commands and tools in the Vectorworks program, along with their location in the workspace. Other useful help sections include "Drawing Structure" on page 155, "Creating Objects" on page 1, "Editing Objects" on page 1, "Viewing the Drawing" on page 2, and "Vectorworks 2015 Keyboard Shortcuts" on page 1.

This table provides the approximate Vectorworks equivalent to many common AutoCAD terms.

AutoCAD	Vectorworks
AC Trim	**Trim** tool
Annotation tools	Dims/Notes tool set
Array (Polar, Rectangular)	**Duplicate Array** command
Attributes	Records, schedule worksheets
Block	Symbol or group
Block command	**Create Symbol** command (or **Group** command)
Break	**Trim** tool, **Connect/Combine** tool
Build Panel	Building Shell tool set
Chamfer 0 0	**Join** command
Circle	**Oval** tool
Construction Line	Guide
Design Center	Resource Browser
Copy	Duplicate
Distance	**Tape Measure** tool
DRAWORDER	**Send** command
Dynamic Dialog, palette	Palette
Entity	Object
Erase	**Clear** command or Delete key
Explode	Symbols and groups can be edited with the **Edit Symbol** or **Edit Group** commands; exploding first (**Decompose** or **Ungroup** commands in Vectorworks) is unnecessary
Extend	**Connect/Combine** tool
Grips	Handles
Insert	**Symbol Insertion** tool
Layer	Class (Vectorworks layers have no equivalent in AutoCAD)

AutoCAD	Vectorworks
Linetype	Line type
MATCHPROP	**Eyedropper** tool
Menu	Workspaces
MLine	**Wall** tool and **Round Wall** tool (Vectorworks walls have a 3D height and additional functionality)
Model space	Design layers
Move	Select and drag the object
MText	Text block
Osnap	SmartCursor cues, Snapping palette
Paper space	Sheet layers
PEDIT	**Compose** command
PLINE	**2D Polygon** tool, **Polyline** tool
Polygon	**Regular Polygon** tool
Properties	Attributes palette and/or the Object Info palette of a selected object
Property Palette	Object Info palette
Scale	**Scale Objects** command
Stretch	Resize by selecting the object and dragging the handles
Toolbar	Tool palette
User Coordinate System	**Snap to Grid** tool and **User Origin** commands
VPORTS, Viewport Windows	Viewports, including sheet layer viewports, design layer viewports, section viewports and detail viewports (Vectorworks Design Series required for certain viewports)
Xref	Workgroup referencing
Zoom All	**Fit to Objects** command
Zoom Extents	**Fit to Page Area** command

Miscellaneous Topics

Cutting Sections

In the Vectorworks Fundamentals workspace, the cutting section commands define a section line through a 3D model, placing the cut section on a new layer and leaving the original model intact.

> Creating section viewports (Vectorworks Design Series required) may be preferable to creating cutting sections. However, the **Cut 2D Section** command and **Cut 3D Section** command can be added to Design Series workspaces if desired.

The **Cut 2D Section** command creates a cross-section, or 2D contour, on the cutting plane. The contour is created by the intersection of the model with an infinite plane passing though the section line. Only the elements that actually intersect the section line are shown.

The **Cut 3D Section** command creates a section with all the 3D geometry that remains on the indicated side of the infinite plane passing through the section line. The elements that intersect the section line, as well as the 3D geometry that exists beyond the line, are shown.

> Place a 2D section, along with a bold line, on top of a 3D section, to show the cutting plane with the section behind it.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cutting 3D Sections
Cutting 2D Sections
Creating Section Viewports

Cutting 3D Sections

The **Cut 3D Section** command (Vectorworks Fundamentals only) cuts a 3D section, or slice, through a 3D model while leaving the model intact. The slice is placed on a new design layer.

To cut a 3D section from a 3D model:

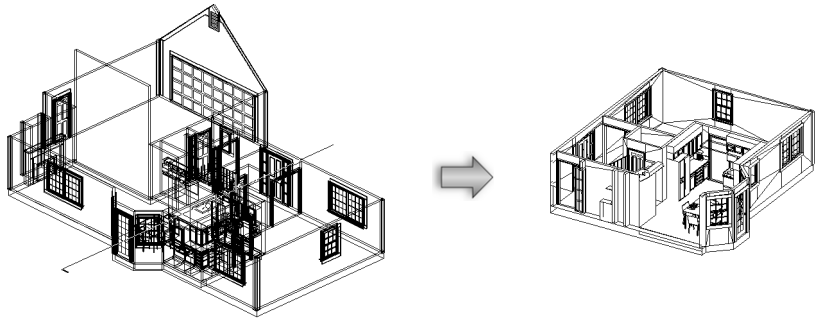
1. Select the 3D model to section.
2. Select **Model > Cut 3D Section**.

   The cursor changes to cross-hairs.
3. Click to set the start of the section. Draw a line across the object to define the section, and then click to set the end of the section.

   When cutting a section while the drawing is in a Plan projection, the cutting plane (and the cut edge of the object) is perpendicular to the active layer plane.

   When cutting a section while in a 3D projection, the cutting plane is perpendicular to the working plane.
4. Click on one side of the line to indicate the portion of the model to keep.

   The Vectorworks program automatically creates a new design layer and places the cut 3D section on it. The original layer remains intact. The new 3D section behaves like any other Vectorworks 3D object.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~



Select the object(s) and select Model > Cut 3D Section; set the section line and click on either side of it to specify the section to keep

The 3D section (rotated and rendered) is created on a new design layer; the original object(s) are left intact

Dimensions and text are 2D objects; therefore, they do not rotate with the cut 3D section.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cutting Sections
Cutting 2D Sections

Cutting 2D Sections

The **Cut 2D Section** command (Vectorworks Fundamentals only) cuts a 2D section, or a slice, from a 3D model without affecting the model. The slice is then placed on a new design layer. For example, to show the profile or a 2D cutaway section of an object in a mechanical 3D drawing, use this command to create the cutaway section in 2D quickly and easily, without affecting the original object.

To cut a 2D section from a 3D model:

1. Select the 3D model to section.
2. Select **Model > Cut 2D Section**.

   The cursor changes to cross-hairs.
3. Click to set the start of the section. Draw a line across the object to define the section, and then click to set the end of the section.

   When cutting a section while the drawing is in a Plan projection, the cutting plane (and the cut edge of the object) is perpendicular to the active layer plane.

   When cutting a section while in a 3D projection, the cutting plane is perpendicular to the working plane.
4. Click on one side of the line to indicate the portion of the model to keep.

   The Vectorworks program automatically creates a new design layer and places the cut 2D section on it. The original layer remains intact.

Layer Linking

Creating Layer Links

Layers are independent of each other. Each design layer has its own scale, view, and render status. In the Vectorworks Fundamentals product, however, a layer link can be created that combines the geometry of several design layers, including referenced layers, onto a single design layer. The linked objects on this design layer display in the same view and scale, and share the same render status. This can then be used to give an accurate depiction of how objects in each layer work together. For example, the various floors of a building can be drawn on separate layers and then linked together into a new layer to form an entire building.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

In the Vectorworks Fundamentals product, consider using viewports instead of layer links, as they provide a better and easier way to present drawings.

In the Vectorworks Design Series products, layer links are being superseded by design layer viewports. For backward compatibility, the **Create Layer Link** command can still be added to any of the Vectorworks Design Series workspaces, and existing layer links can still be viewed and edited. For information on design layer viewports, see "Creating Design Layer Viewports" on page 1620.

The layer link is created on a new design layer that contains links to the existing design layers of the drawing. 3D objects on selected layers are automatically linked; 2D planar or screen objects can be displayed in the layer link. Once the layer link is created, updates to the design layers are automatically reflected on the linked layer when a screen redraw occurs. However, this updating occurs only in one direction; any new objects or details added to the linked layer will not appear in any other layers. Linked objects cannot be edited on the linked layer; they must be edited on their source layer.

To create a layer link:

1. Create a new layer, and then make it the active layer.

   This layer shows objects on all linked layers and any changes made to them.

2. Select **View > Create Layer Link**.

   The Create Layer Link dialog box opens; the layer being linked to (the currently active layer) is not listed.

Click to show/hide the parameters.

Parameter	Description
Layers list	Lists the existing design layers; sheet layers and the active design layer are not displayed. Referenced layers display in italics.
Display Planar Objects	Select to display 2D planar objects, when the layer link is in a view other than Top/Plan
Project Screen Objects	Select to display 2D objects associated with the screen plane, when the layer link is in a view other than Top/Plan

3. Select the design layers to be linked from the list of existing layers.
4. Click **OK**.

Linked layers are locked objects. To unlock a linked layer, select **Modify > Unlock**. Double-click on an item in the layer link to return to its source layer and edit it.

To project 2D planar or screen objects after a layer link has been created, select and then unlock the layer link object. Select the options in the Object Info palette.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cropping Layer Links
Creating Sheet Layer Viewports
Creating Design Layer Viewports
Converting Layer Links

Cropping Layer Links

Layer link objects can be cropped in a similar manner to viewports (see "Cropping Sheet Layer or Design Layer Viewports" on page 1652), although the area outside of the crop cannot be displayed as it can for a sheet layer or design viewport. When cropped, only a portion of the layer link displays; increase the scale of the layer to create a detailed view. Layer links with workgroup-referenced layers can also be cropped.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

To crop a layer link:

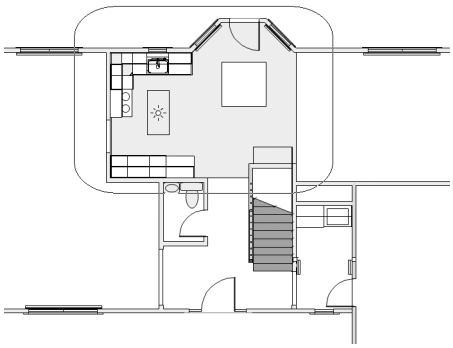
1. Select an existing layer link.
2. Unlock the layer link by selecting **Modify > Unlock**.
3. Click **Edit Crop** from the Object Info palette to enter the Layer Link Crop mode.

   Alternatively, right-click (Windows) or Ctrl-click (Mac) on a layer link and select **Edit Crop** from the context menu.

   A colored border around the drawing window indicates that you are in an editing mode. The **Exit Crop** command becomes available from the **Modify** menu, and the **Exit Layer Link Crop** button displays in the top right corner of the drawing window.
4. Create a 2D object such as a rectangle, circle, or polyline. The 2D object must define an area; a 2D line, for example, cannot be used. Position the 2D object to delimit the new crop display area. The fill of a cropping object is always None; however, the pen style can be set from the Attributes palette while in Edit Crop mode. Set the pen style to None (or the crop object class to invisible) to make the crop object invisible. Move and resize the 2D object as needed.

   To view other objects while in Edit Crop mode, select **Show other objects while in editing modes** on the Display tab of the Vectorworks preferences (see “Vectorworks Display Preferences” on page 50).

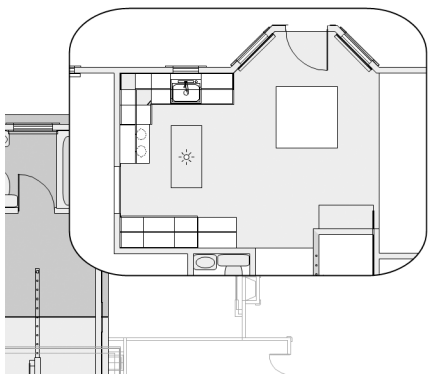
   Use the **Flyover** tool to adjust the view as necessary (see “Flyover” on page 1144).



5. Click **Exit Layer Link Crop**, or select **Modify > Exit Crop** to return to the drawing.

   The cropped layer link is displayed. In the Object Info palette, the crop status has changed to **Yes**.

   By increasing the scale of the layer with the layer link, and making other layers visible, a floor plan can be displayed (original design layer) along with a detailed view of the floor plan (zoomed in, cropped layer link).



To change, replace, or delete the layer link crop, select the cropped layer link and then select **Edit Crop** from the Object Info palette to re-enter crop mode. Click **Exit Layer Link Crop**, or select **Modify > Exit Crop** to return to the drawing.

The entire layer link is displayed if a viewport of a cropped layer link is created.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Layer Links

Converting Layer Links

In the Vectorworks Design Series products, layer links are being superseded by design layer viewports. Because users of the Vectorworks Fundamentals product cannot create design layer viewports, and users of the Vectorworks Design Series products cannot create layer links, it occasionally may be necessary to convert a layer link into a viewport, or to convert a design layer viewport into a layer link.

- To convert a layer link into a design layer viewport, first unlock the layer link. Then, right-click (Windows) or Ctrl-click (Mac) on the layer link, and select **Convert to Viewport** from the context menu.
- To convert a design layer viewport into a layer link, select the viewport and then select **Modify > Decompose**. The Object Info palette changes to indicate that the object is now a group. Select **Modify > Ungroup** to create a layer link.

D Records and Schedules

Some architectural elements (doors and spaces, for example) have pre-defined data that is automatically attached to them as they are created. Pre-formatted schedules are available for reporting this data, as well as data from other elements of the Vectorworks Architect, Landmark, and Spotlight products. (For information on choosing a preformatted schedule in Vectorworks Landmark and Spotlight, see “Creating Schedules Automatically” on page 1318.)

Typically, custom data records and schedules are created and edited with the Resource Browser. The Record and Schedule Management suite provides an alternate method for managing data in architectural documents.

For information about using the Resource Browser to create and edit record formats and schedules (worksheets), see “Record Formats” on page 262 and “Creating Worksheets” on page 1315. Record formats and schedules created in this way cannot be used with Vectorworks Architect product’s Record and Schedule management suite, however.

The records and schedules suite includes the following commands:

- **VA Records and Schedules:** Establishes default formats for records that can be attached to objects, and for schedules that can be generated from those records
- **VA Create Record:** Adds selected record formats to the current file
- **VA Create Schedule:** Adds selected schedules to the current file

By default, records and schedules created with **VA Records and Schedules** are added to the VA Defaults project preference set, and new files use that VA Defaults set. If additional sets of records and schedules are needed, create a new set of preferences and select that set before setting up a new drawing file. See “Working with Project Preference Sets” on page 1866 for more information.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Records and Schedules
Creating Records
Creating Schedules

Defining Records and Schedules

Use the **VA Records and Schedules** command to create and modify a set of record format and schedule definitions that can be used in any file.

After defining record formats and schedules, make them available in the current file using the **VA Create Record** and **VA Create Schedule** commands.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Editing Record Definitions

To edit a record definition:

1. Select **Tools > Reports > VA Records and Schedules**.

   The Records and Schedules dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Records	Lists currently defined record formats
Add	Adds a new record to the current preference set
Remove	Deletes the selected record
Edit	Opens the selected record for editing
Schedules	Lists currently defined schedule formats
Add	Adds a new schedule to the current preference set
Remove	Deletes the selected schedule
Edit	Opens the selected schedule for editing

2. Select the record to edit.
3. Click **Edit** in the Records section of the Records and Schedules dialog box.

   The Record Formats dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Record Fields	Lists the current fields of the selected record
Add	Creates a new record field
Remove	Deletes the selected record field (cannot be undone)
Edit	Opens the selected record field for editing

4. Click **Add** to add a new record field or click **Edit** to edit the selected field.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

The Edit Field dialog box opens. Add or edit the field information.

Click to show/hide the parameters.

Parameter	Description
Name	Specifies the name of the selected record field
Default	Enter a default value for the field, if desired
Type	Specifies the type of field: Integer, Boolean, Number, or Text

5. Click **OK** to close the Edit Field dialog box.
6. In the Record Formats dialog box, continue to modify, add, or remove record items.
7. Click **OK** to close the Record Formats dialog box, and then click **Done** to close the Records and Schedules dialog box.

   Once a record definition has been updated, instances of the new record format cannot co-exist in the same file with instances of the old record format. If an object is inserted into a drawing with a record format that differs from the current record of the same name, a dialog box opens requesting which format to retain. Records in the un-retained format will be purged, and their information will be lost. For this reason, it is recommended that any changes be made to record formats prior to beginning a project.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Records and Schedules

Adding Record Definitions

To add a record definition:

1. Select **Tools > Reports > VA Records and Schedules**.

   The Records and Schedules dialog box opens.
2. Click **Add** in the Records section of the Records and Schedules dialog box.

   The Enter Text dialog box opens. Enter the name of the new record.
3. Click **OK**.

   The Record Formats dialog box opens, listing the default fields of the new record; only the **Add** button is enabled.
4. Click **Add** to add a new field to the current record definition. Enter a **Name** and **Default** value for the field. Select the field **Type** from the list.
5. Click **OK** to close the Edit Field dialog box.

   The new field displays on the Record Formats dialog box. If more fields are needed, repeat steps 4 and 5.
6. Click **OK**.

   The new record and its fields are added to the record definition list.
7. Click **Done** to close the Records and Schedules dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Records and Schedules

Removing Record Definitions

To remove a record definition:

1. Select **Tools > Reports > VA Records and Schedules**.

The Records and Schedules dialog box opens.

2. Select the record to remove.
3. Click **Remove** in the Records section of the Records and Schedules dialog box. This operation cannot be undone; click **Yes** to confirm that the record should be removed.

   The record definition is removed from the record list.
4. Click **Done** to close the Records and Schedules dialog box.

Defining Records and Schedules

Editing Schedule Definitions

The Schedule definitions detail the column heading text, width, and border attributes for each record field being reported.

To edit a schedule definition:

1. Select **Tools > Reports > VA Records and Schedules**.

   The Records and Schedules dialog box opens.
2. Select the schedule to edit.
3. Click **Edit** in the Schedules section of the Records and Schedules dialog box.

   The Edit Schedule Format dialog box opens. Specify the schedule appearance by selecting the column order and formatting.

Click to show/hide the parameters.

Parameter	Description
Schedule Information	Select the record to format, defining the data to be reported in the schedule
Available Record Fields	Select the record field(s) to include in the schedule
Schedule Columns	Displays the fields that will be included in the schedule
Add >	Moves the selected available record field into the Schedule Columns list
< Remove	Deletes a selected field from the Schedule Columns list
Move Up	Moves the selected field up in the Schedule Columns list order
Move Dn	Moves the selected field down in the Schedule Columns list order
Heading	Enter a title for the selected Schedule Column item, if desired. The title displays in row 2 of the column corresponding to the field.
Sub-Heading 1	Enter a subheading for the selected Schedule Column item if desired. The title displays in row 3 of the column corresponding to the field.
Sub-Heading 2	Enter a subheading for the selected Schedule Column item, if desired. The title displays in row 4 of the column corresponding to the field.
Width (chars)	Specifies the width, in characters, for the selected Schedule Column item
Borders	Specifies the column border display for the selected Schedule Column
Options	Opens the Schedule Format Options dialog box; proceed with step 4

4. Click **Options** to edit the schedule font style and size.

The Schedule Format Options dialog box opens; the current font type and size settings for the heading, subheadings, and body display.

5. If different font settings are desired, click the appropriate **Change** button to open the Format Text dialog box and modify the font type and size for the heading, subheadings, and body of schedule items. (See "Formatting Text" on page 389.)
6. Click **OK** to close the Schedule Format Options dialog box, and then click **OK** to close the Edit Schedule Format dialog box.
7. Click **Done** to close the Records and Schedules dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Records and Schedules

D Adding Schedule Definitions

To add a schedule definition:

1. Select **Tools > Reports > VA Records and Schedules**.

   The Records and Schedules dialog box opens.
2. Click **Add** in the Schedules section of the Records and Schedules dialog box.

   The Enter Text dialog box opens.
3. Enter the name for the new schedule.
4. Click **OK**.

   The Edit Schedule Format dialog box opens. For details on the Edit Schedule Format dialog box parameters, see "Editing Schedule Definitions" on page 1862.
5. Select the record that defines the data to be reported in the new schedule, and then add the desired fields to use for the schedule.
6. Select a schedule column, and then enter the column formatting information. If desired, click **Options** and set the font formatting for the column, and then click **OK**.
7. Click **OK** to save the new schedule.

   The new schedule definition is added.
8. Click **Done** to close the Records and Schedules dialog box.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining Records and Schedules

A Removing Schedule Definitions

To remove a schedule definition:

1. Select **Tools > Reports > VA Records and Schedules**.

   The Records and Schedules dialog box opens.
2. Select the schedule definition to remove.
3. Click **Remove** in the Schedules section of the Records and Schedules dialog box. This operation cannot be undone; click **Yes** to confirm that the schedule should be removed.

   The schedule definition is removed from the schedule list.
4. Click **Done** to close the Records and Schedules dialog box.

Defining Records and Schedules

Creating Records

The **VA Create Record** command adds record formats created with the **VA Records and Schedules** command to the current file. Once the records are added, they can be attached to items in the drawing using the **ID Label** tool. They can also be attached by selecting an object and then selecting the check box next to the record format in the Data tab of the Object Info palette.

To create a record:

1. Select **Tools > Reports > VA Create Record**.

   The Create Record dialog box opens, listing the available records that can be added to the file.
2. Select the record to add to the file.
3. Click **OK** to add the specified record to the file.

Creating Schedules

The **VA Create Schedule** command generates worksheets for the current file. Any schedule definitions that were created with the **VA Records and Schedules** command display in the list, as well as several pre-formatted schedules provided as default content with the Vectorworks Architect product.

> The **Display default content** option must be selected on the Session tab of Vectorworks preferences to see the pre-formatted schedules. (See “Session Preferences” on page 52.)

Pre-formatted schedules include:

- Diffuser Report
- Door Hardware Legend
- Door Schedule
- Door Schedule w/Images
- Drawing List
- Equipment Schedule
- Existing Tree Schedule
- Existing Tree Schedule w/Images
- Irrigation Head Schedule
- Irrigation Line Schedule
- Hardscapes Budget UK
- Hardscapes Budget US
- Objects with IFC Entity
- Objects with IFC Entity - Specific
- Objects without IFC Entity
- Plant List - Basic
- Plant List - Basic w/Images
- Plant List - Colors
- Plant List - Colors w/Images
- Plant List - Costing
- Plant List - Costing w/Images
- Plant List - Simple
- Plant List - Simple w/Images
- Plant List - Types
- Plumbing Schedule
- Roof Face Area
- Room Finish Schedule
- Wall Area
- Wall Style Report
- Wall Style Report w/Images
- Window Schedule
- Window Schedule w/Image
-

To generate a schedule:

1. Select **Tools > Reports > VA Create Schedule**.

   The Create Schedule dialog box opens, listing the schedules that can be added to the file.
2. Select the desired schedule to be created. Enter a new **Schedule Title** if desired. Select **Place worksheet on drawing** to place the worksheet on the drawing for printing.

3. Click **OK**.

   Each schedule has a record format associated with it. If there is a difference between the record definition currently in the file and the record definition in the current preference set, a notice displays with the option to continue or fix the record.

   If the selected schedule already exists in the file, a warning dialog box opens. Select whether to replace or rename the new schedule (some schedules also have a recalculate option), and click **OK**.

4. Click on the drawing area where the top left corner of the schedule is to be located.

   The worksheet opens, displaying the schedule information. If the option to place the worksheet on drawing was selected, the worksheet is included on the drawing.

   If the "on drawing" worksheet is accidentally deleted, select the worksheet name from the Resource Browser and select **Resources >Worksheet On Drawing**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a Room Finish Legend

Creating a Room Finish Legend

There can be dozens of room finishes assigned within a project file. To obtain a detailed report of all room finish data, first generate a room finish schedule as described in "Creating Schedules" on page 1864. Then generate a detailed legend showing a full description of the finishes using the **Create Rm Finish Legend** command. This command can only be used after room finishes have been assigned to space objects.

To generate a room finish legend:

1. Select **Tools > Reports > Create Rm Finish Legend**.

   The Create Room Finish Legend dialog box opens.

Click to show/hide the parameters.

Parameter	Description
Create legend as text block	Adds the legend to the drawing as a text block
Label Indent	Sets the offset from the location title to the item title
Note Indent	Sets the offset from the location title to the item description
Section Spacing	Sets the text spacing between each section
Note Spacing	Sets the text spacing between each item description
Bold Section Titles	Displays the location titles in bold face text
Create legend as worksheet	Adds the legend to the drawing as a worksheet
Worksheet Name	Sets the name of the worksheet
Place worksheet in document	Places the worksheet on the drawing at the next mouse click
Add row between sections	Adds a worksheet row between each location section
Bold section titles	Displays the location titles in bold face text

2. Select whether the legend should be displayed as a text block or as a worksheet. See "Creating Text Blocks" on page 384 and "Creating Worksheets" on page 1315.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Display Method	Description
Text block	Define the area for the legend to occupy. Click to set the top left corner of the text block, and then click to set the bottom right of the text block. The text legend is generated within the defined rectangle.
Worksheet	The worksheet is created with the legend information. If the **Place worksheet in document** option is selected, click on the drawing to set the top left corner of the worksheet.

3. Click **OK**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating Schedules

Working with Project Preference Sets

In most cases, the default project preference set (VA Defaults) is the only set needed. If additional sets of records and schedules are needed, create a new set of preferences and select that set before setting up a new drawing file.

A project preference set is a grouped configuration of record, schedule, and library information that is used by a drawing file. A project preference set can be customized for particular projects (residential or commercial, for example) or different types of clients (restaurant or retail store, for example). Different record and schedule formats can be assigned, as well as information in room finish libraries and door hardware libraries. These custom configurations are saved in external text files. The **VA Set Project Prefs** command assigns which set of external files to use within the current drawing. The settings files used by these commands, taken together, are called a "project preference set." Dialog boxes in the Vectorworks Architect product's Record and Schedule management suite display the name of the current preference set in the title bar.

Because the preference sets are stored as external files, they can be shared among many users to comply with office standards. The preference set folders can be copied to the same location for each user, or shared on a network location. Either select the **VA Set Project Prefs** command to designate the preference set location, or copy the following text file to each computer:

[Vectorworks]\Plug-Ins\VW_Arch\Data\VA2_Preference_Set_Paths.txt

See "Project Preference Sets" on page 1897 for information on the files contained in the preference set folders.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Selecting a Project Preference Set
Creating a New Project Preference Set

Selecting a Project Preference Set

The default preference set is named "VA Defaults," and its files are located in [Vectorworks]\Plug-Ins\VW_Arch\Data\Prefs_Def.

To use a different project preference set, select it when setting up the file. The preference set selection is saved with the file. Changes to VA Records and Schedules, the Door Hardware Library, or the Room Finish Library are saved in the currently selected project preference set.

> When using template files for different types of projects, make a preference set selection before saving the template.

To select a project preference set:

1. Select **Tools > Options > VA Set Project Prefs**.

   The VA Preference Sets dialog box opens. The listed sets are available for use in the file.

Click to show/hide the parameters.

Parameter	Description
Preference set list	Lists project preference sets that are available for use in the file
Add	Adds a project preference set to the list; specify the set location and name
Remove	Deletes the selected project preference set association (the file is not deleted, but cannot be selected for this project)
Edit	Changes the project preference set name or location

2. Select the project preference set to activate.
3. Click **Done**.

   The selection “VA Defaults” cannot be deleted or edited; it contains the information that is used if no project preference set is designated or if there is an error finding the designated set.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Creating a New Project Preference Set

Creating a New Project Preference Set

While the Vectorworks Architect product installs with six project preference sets, it supports the use of an unlimited number. To add preference sets beyond the default options, manually create additional preference folders and then add the set to the list of defined sets using the **VA Set Project Prefs** command.

The project preference set folder must contain the following files for correct operation:

- Door Hardware Library.txt
- Equipment Record.txt
- Equipment Schedule.txt
- Plumbing Fixt Record.txt
- Plumbing Schedule.txt
- VA2 Records and Schedules.txt
- VA_Project_Set.txt

If additional records and schedules are defined using the **VA Records and Schedules** command, their data files are stored in the project preference set folder.

Adding a Project Preference Set

After creating a project preference set, add it to the list of project preference sets available:

1. Select **Tools > Options > VA Set Project Prefs**.
2. The VA Preference Sets dialog box opens. Click **Add**.

   The Add Preference Set dialog box opens. Specify the name and location of the project preference set.

Click to show/hide the parameters.

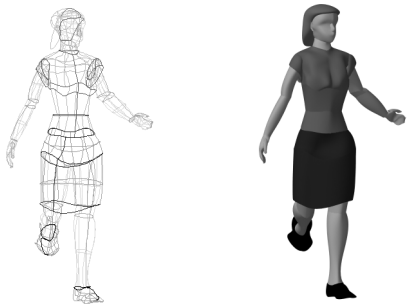
Parameter	Description
Path	Enter or browse for the path to the project preference set folder
Name	Specifies a name for this project preference set

3. Click **OK** to return to the VA Preference Sets dialog box. To use the new project preference set, select it from the list and click **Done**.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Records and Schedules

Creating a Human Figure

A 3D human figure object can be inserted for a realistic addition to the drawing. Constructed of NURBS surfaces for a high-quality model, the object contains a wide variety of parameters for full control over the figure type, position, and appearance.



An entourage/figures object library also contains pre-configured human figure objects in a variety of poses and figure types (see “Resource Libraries” on page 219).

Several figures can increase the time required to render the model in 3D.

Inserting the Figure and Setting Parameters
Setting Figure Attributes
Specifying a Custom Figure Position

Inserting the Figure and Setting Parameters

To insert and configure a human figure:

1. Click the **Human Figure** tool from the Visualization tool set.
2. Click to place the human figure in the drawing.
3. The human figure parameters can be modified from the Object Info palette.

Click to show/hide the parameters.

Parameter	Description
Height	Specifies the figure height, in the current unit
Body Type	Select the figure body type (Slight/Petite, Average, or Large)
Figure Type	Select the type of human figure (Man, Woman, Boy, or Girl)
Posture	Select the figure position; Custom is displayed when the figure posture has been adjusted in the Figure Custom Position dialog box (see “Specifying a Custom Figure Position” on page 1870)
Hand Height	For the Standing (shaking hands) posture, specifies the height of the hands, in the current unit
Attire (Top)	Select the clothing for the top of the figure

Parameter	Description
Attire (Bottom)	Select the clothing for the bottom of the figure A dress overrides the selected top attire
Footwear	Select the figure's footwear, or select None if the figure is barefoot
Hair	Select the figure's hairstyle
Jacket	Places a jacket on the figure
Stockings	Places stockings on the figure
Tie	Places a tie on the figure
Set Attributes	Click to set the figure attributes more specifically (see "Setting Figure Attributes" on page 1869)
Custom Position	Click to set a custom figure position (see "Specifying a Custom Figure Position" on page 1870)

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Setting Figure Attributes
Specifying a Custom Figure Position

Setting Figure Attributes

Figure attributes can be set precisely to control the figure appearance.

To set figure attributes:

1. Select the desired figure.
2. In the Object Info palette, make the initial selections for the figure.
3. Click **Set Attributes** from the Object Info palette.

   The Human Figure Attributes dialog box opens, with the figure set to the current Object Info palette parameters. Select each desired model part and then set its attributes.

Click to show/hide the parameters.

Parameter	Description
Preview	Displays a preview of the figure in the specified posture, with a sample of the selected texture and attributes
Rotate Left/Right	Rotates the figure preview 90 degrees to the left or right
Render	Select a rendering mode for the preview figure. If **Automatically Render** is not selected, click **Render** after each attribute change to view a rendered preview.
Automatically Render	Renders the preview figure automatically, according to the selected mode, after an attribute change (depending on the selected rendering mode and the inclusion of attributes such as textures, this can take some time)
Model parts	Select each figure attribute to configure, and then set its parameters
Attributes	
Class	Select a class to apply to the attribute from the list of classes in the file, or select Container to apply the object's class to the selected attribute
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Fill Style	Select Class to use the class style for the selected attribute, choose None to apply no fill, or choose Solid to apply a solid color and then click on the color box to select the fill color
Texture (Renderworks required)	Sets the texture parameters for the selected attribute from either the default content or the current file's content; see "Resource Libraries" on page 219
Texture	Select the texture to apply from either the default content or the textures in the current file, or select Class to apply the class texture; select None to apply no texture
Map Type	Select the texture map type
Radius	For sphere and cylinder maps, sets the texture radius; the default radius is the same as the 3D object radius. Increasing this value reduces the size of the texture on the object.
Repeat Horizontally/ Vertically	Repeats the texture in a horizontal and/or vertical direction
Scale Factor	Determines the texture size when projected onto the object; either enter a scale value or use the slider to change the scale
Horizontal/ Vertical Offset	Sets the start location of the texture horizontally and vertically
Rotation	Sets the angle of texture rotation; either enter a rotation value from 0 to 360 degrees or use the slider to change the rotation angle

For more information on texture parameters and mapping, see "Applying and Mapping Textures" on page 1527.

4. Click **OK** to apply the attributes to the figure.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting the Figure and Setting Parameters
Specifying a Custom Figure Position

D Specifying a Custom Figure Position

If the **Posture** selection is not sufficient to position the figure exactly as desired, a custom figure pose can individually position the figure's head, body, arms, and legs.

Use the **Rotate** tool to set the overall position of the figure.

To specify a custom figure pose:

1. Select the desired figure.
2. In the Object Info palette, make the initial parameter selections for the figure.
3. Click **Custom Position** from the Object Info palette.

   The Figure Custom Position dialog box opens, with the figure set to the current Object Info palette parameters. Select the body part tab to change, and then set the part parameters by dragging the slider for the desired settings.

   Use slight slider adjustments for best results.

Click to show/hide the parameters.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Preview	Displays a preview of the figure with the specified figure pose
Rotate Left/ Right	Rotates the figure preview 90 degrees to the left or right
Render	Select a rendering mode for the preview figure. If **Automatically Render** is not selected, click **Render** after each position change to view a rendered preview.
Automatically Render	Renders the preview figure automatically, according to the selected mode, after a position change (depending on the selected rendering mode and the inclusion of attributes such as textures, this can take some time)
Revert	Returns the figure to its original position
Head	Specifies the head parameters
Angle	Indicates where the head is positioned along a vertical axis (looking “up” or “down”). Move the slider left to move the head up, and move the slider right to look down.
Tilt	Specifies how much the head tilts to the left or right; move the slider left to tilt over the right shoulder, and move the slider right to tilt over the left shoulder
Turn	Sets the amount that the head turns to the left or right; move the slider right to look over the left shoulder, and move the slider left to look over the right shoulder
Body	Specifies the body trunk parameters
Hip Bend	Sets the bend of the lower torso and legs, by setting the angle between the hip and thigh. Move the slider right to set the angle towards the back of the body, and move the slider left to set the angle towards the front of the body
Waist Bend	Indicates the bend of the upper torso; move the slider right to bend forwards, and left to bend backwards
Waist Tilt	Indicates how much the upper torso tilts to the left or right; move the slider right to tilt towards the left, and move the slider left to tilt towards the right
Waist Twist	Specifies how much the upper torso twists to the left or right; move the slider right to rotate towards the left, and move the slider left to rotate towards the right
Arms	Specifies the arm parameters
Arm Selector	For each parameter, select the arm for the position change, or select Both Arms to mirror the position change on both sides of the body
Arm Rotation	Specifies the arm position at the front or back of the body; move the slider right to move the arm behind the body, and move the slider left to move the arm in front of the body
Arm Twist	Specifies the arm position as it twists along the arm axis; move the slider right to rotate the arm counter-clockwise, and move the slider left to rotate the arm clockwise
Arm Angle	Sets the up and down position of the arm; move the slider left to move the arm away from the body, and move the slider right to move the arm towards the body (depending on which arm moves, this may be reversed)
Elbow Bend	Specifies the amount that the elbow is bent; normally, move the slider left to bend the elbow in a natural direction
Wrist Twist	Sets the rotation of the wrist; move the slider left to twist the wrist clockwise and to the right to twist the wrist counter-clockwise

Parameter	Description
Hand Twist	Sets the flex of the hand; move the slider to the left to flex the hand towards the front of the body, and move the slider to the right to flex the hand towards the back of the body
Legs	Specifies the leg and foot parameters
Leg Selector	For each parameter, select the leg for the position change, or select Both Legs to mirror the position change on both sides of the body
Leg Rotation	Specifies the leg position at the front or back of the body; move the slider right to move the leg behind the body, and move the slider left to move the leg in front of the body
Leg Twist	Specifies the leg position as it twists along the leg axis; move the slider right to rotate the leg counter-clockwise, and move the slider left to rotate the leg clockwise
Leg Angle	Sets the up and down position of the leg; move the slider left to move the leg away from the body, and move the slider right to move the leg towards the body (depending on which leg moves, this may be reversed)
Knee Bend	Specifies the amount that the knee is bent; normally, move the slider left to bend the knee in a natural direction
Foot Rotation	Sets the rotation of the foot; move the slider left to twist the foot clockwise and to the right to twist the foot counter-clockwise
Foot Flex	Sets the flex of the foot; move the slider to the left to flex the foot towards the front of the body, and move the slider to the right to flex the foot towards the back of the body

4. Click **OK** to apply the custom position to the figure.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Inserting the Figure and Setting Parameters
Setting Figure Attributes

Exporting for DOE-2 Requirements

With rising energy costs and stricter regulations on energy usage in commercial buildings, it is becoming increasingly important to be able to estimate the energy cost implications of design strategies, especially during the schematic design phase. Energy usage programs develop these estimates. The Vectorworks Architect product can export the building geometry to a file in the format read by the DOE energy analysis engine.

To create a DOE-2 export file:

1. The initial building space planning should be complete, with layers for all the building levels and spaces on those layers. Exterior walls are also required; create these automatically with the **Create Walls from Spaces** command, or manually.

   Windows and doors should also be inserted into the walls where appropriate.

2. Select **AEC > Space Planning > Export DOE-2**.

   The Export DOE-2 dialog box opens.

Click to show/hide the parameters.

Parameter	Description
File Name	Enter the path and file name for the DOE-2 file, or click **Browse** to specify the location and file name
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Confirm Layer Heights	Displays the floor elevation and ceiling heights as specified in the Layers Setup dialog box; these values can be edited
Up/Down	Scroll up and down through the list of layers (available when there are more than five layers)

3. Click **OK** to create the DOE-2 text file. Send the file to your DOE consultant for analysis.

   The relationship between Vectorworks entities and their corresponding BDL language entities is shown in the following table.

Vectorworks	DOE
Distance from layer Z to next highest layer Z	Floor height
Layer delta-Z	Space height
Description field in wall record	Construction type
Wall record R-value	Construction type U-Factor
Window vs. door record	Window vs. door

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Generating an INP File from a DXF File

Generating an INP File from a DXF File

If .INP files need to be generated from DXF files for analysis with the DOE-2 engine, several steps are required to properly import and format the file in the Vectorworks Architect product prior to exporting it.

X-REFs and Layers

AutoCAD building plans are often broken up into several files, which are then cross-referenced (x-ref) to a master file. Prior to importing into a Vectorworks file, all of these files must be pulled into the master file. Essentially, all of the information about a building must be present in one file for the export to recognize the relationships among adjacent building levels and spaces on each of those floors. See “DXF/DWG and DWF File Import” on page 1716.

Vectorworks classes are roughly equivalent to AutoCAD layers. When importing the DXF file, the layers should be imported as Vectorworks classes. The Vectorworks program allows the specification of the base elevation (Z value) of layers; therefore, Vectorworks layers are typically used to separate the different building levels in the drawing. Once these layers have been established with the appropriate Z values, it is much easier to draw on the layers without having to specify base heights of each object.

The **Export DOE-2** command uses the layer Z values to determine the Z value of each floor in the INP file. After importing a DXF file, first create a layer for every building level, and then assign a Z value to that layer. In addition, assign a delta Z value. In a normal drawing created with the Vectorworks Architect product, the delta Z controls the default height (at object creation time) of all walls on that layer. The **Export DOE-2** command uses the delta Z as the space-height.

To create a layer for a building level, select **Tools > Organization**. On the Design Layers tab, create and set properties for the building level layer. See “Creating Layers” on page 162 and “Setting Design Layer Properties” on page 165.

Defining Spaces

Once the layers have been defined, define the spaces. Polygons may already exist for all the spaces; if so, convert the polygons to spaces with the **Create Spaces from Polys** command (see “Space Settings” on page 406).
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Walls

The Vectorworks program has a dedicated object type for walls. A Vectorworks wall is a complex object type which is a combination of lines, arcs (in the case of rounded walls), polygons/polylines, and fill patterns. The Vectorworks Architect product adds more functionality to wall objects by attaching a record with a description field.

The **Export DOE-2** command checks for wall objects along the perimeter of the spaces in the drawing. If walls are found, it checks for a wall record. If this is found, the description field is exported as the construction type for that segment. If no wall record is present, the command exports the wall's class name as the construction type. If it does not find a wall object on any particular space vertex, a default construction type for the wall on that vertex is assigned. The default construction types are "Default Exterior Wall" construction and "Default Interior Wall" construction.

If the walls in the original DWG file were drawn with multi-lines, importing the multi-lines as Vectorworks walls is recommended. If the original draftsperson did not use multi-lines, then the walls must be manually drawn. The walls must be on the same layer as their respective space objects. In other words, first floor walls must be on the same layer as the space objects for the first floor.

Windows and Doors

The **Export DOE-2** command determines the difference between windows and doors on the basis of the attached record. If the symbol has a door record attached, the symbol is exported as a door. If it has a window record attached to it, or no record at all, it is exported as a window.

To export windows and doors, walls are required. Inserting windows and doors into Vectorworks walls is much easier than creating them mathematically within the DOE-2 interface. Vectorworks walls normally "contain" the windows and doors, essentially as nested objects. The associativity of windows and doors with their parent walls allows the **Export DOE-2** command to recognize which windows/doors are associated with which walls, and export them to the appropriate place in the INP file. Ensure that when inserting the windows and doors into the walls, they create a wall break. Otherwise, they are not considered associated with the wall.

AutoCAD blocks can be used, which convert to Vectorworks symbols during the DXF import. The length of the window/door, parallel to the wall, is exported as the width. For windows, the elevation of the lowest part of the window is exported as the Y value.

Cam Design

The cam design tool can be animated and generates cam data worksheets and diagrams.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cam Template

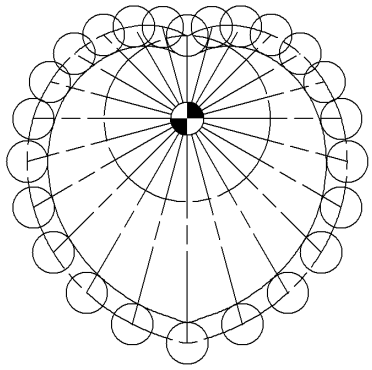
To use the cam design tool, open a new file and select the Cam (Imperial).sta or Cam (Metric).sta template.

To open the cam template:

1. Select **File > New**.

   The Create Document dialog box opens.

2. Select **Use document template**, and choose the Cam (Imperial).sta or Cam (Metric).sta template.
3. Click **OK**. The new file opens, with a cam inserted by default. The file contains Cam and Diagrams layers.
~~~~~~~~~~~~~~~~~~~~~~~~~~~~



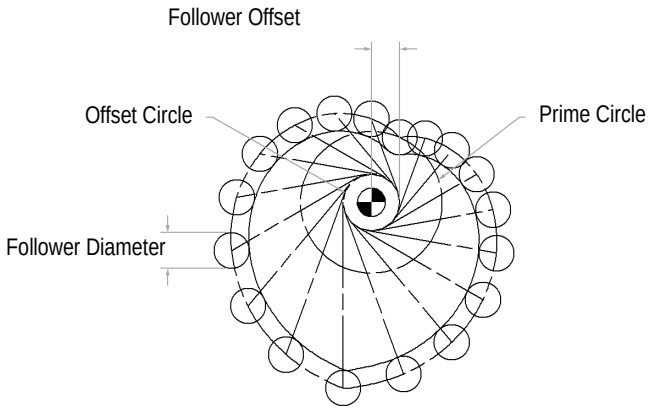
The recommended procedure to include a cam in a Vectorworks drawing is to copy and paste it into the drawing, and convert it to a group by selecting it and selecting **Modify > Convert > Convert to Group**.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cam Properties
Cam Data Worksheets
Cam Diagrams

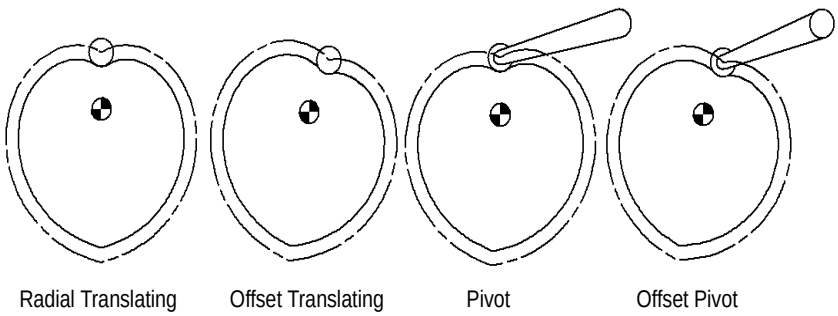
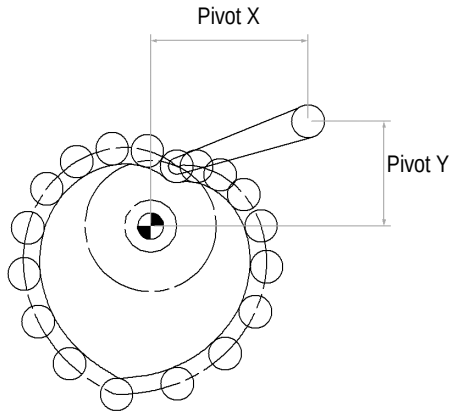
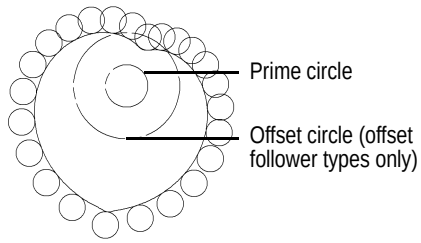
Cam Properties

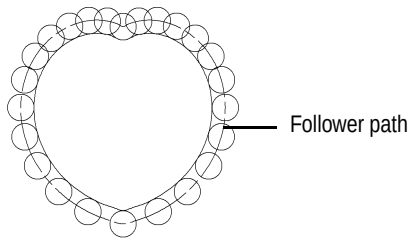
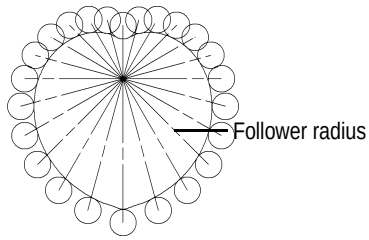
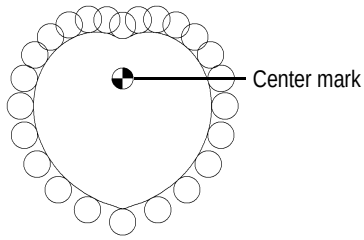
The cam object is a point object located in the Cam (Imperial).sta and Cam (Metric).sta templates. Cam properties can be edited in the Object Info palette.



Click to show/hide the parameters.

Parameter	Description
Prime Circle Radius	Specify the prime circle radius
Cam Type	Select plate or groove
Outside Dia.	Enabled if Groove is selected in **Cam Type**. Enter the outside diameter for groove cam types.
Cam Drawing Incr. (Deg.)	Enter the cam drawing increment, in degrees; this parameter affects how the cam is drawn. The larger the value, the less smoothly the cam is drawn.
Cam Speed (rpm)	Specify the cam rotation in revolutions per minute (rpm); this value is used to calculate the velocity and acceleration of the cam follower
~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Parameter	Description
Follower Type	Select one of the follower types 
Follower Dia.	Enter the follower diameter
Follower Offset	Enabled if Offset Translating Roller or Offset Pivot Roller is selected in **Follower Type**. Indicate the distance between the center of the cam and the follower center for the offset follower types.
Pivot X/Y (from cam center)	Enabled if Pivot Roller is selected in **Follower Type**. Specify the x and y distance from the cam center to the pivot arm for pivot roller follower types. 
Draw Followers	Select to display the followers
Follower Incr. (Deg.)	Determine at what angles to display followers by entering the follower increment in degrees (to show only the actual follower, enter 360). If a segment of the cam ends on an angle that does not correspond to the follower increment, the additional follower is also displayed.
Draw Prime and Offset Circles	Select to display both the prime and offset circles 

Parameter	Description
Draw Follower Path	Select to display the follower path 
Draw Radii	Select to draw a radius from each follower to the center mark 
Draw Center Mark	Select to draw a center mark 
Data Incr. (Deg.)	Specify the increment at which the displacement, velocity, and acceleration of the cam follower are calculated for the worksheet and text file
Define Cam Profile	Click to specify the profile of each cam segment
Write Data to Worksheet	Click to write the results (displacement, velocity, and acceleration) to the worksheet named Cam Data, using the increment specified by **Data Incr.**
Write Data to Text File	Click to write the results (displacement, velocity, and acceleration) to a text file using the increment specified by **Data Incr.**
Animate	Click to animate the cam
Reset Cam to 0	Click to position the cam back to zero degrees; for more information, see “Resetting the Cam to Zero Degrees” on page 1878

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining the Cam Profile
Animating Cam Movement
Cam Template
Cam Data Worksheets
Cam Diagrams
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Defining the Cam Profile

Specify the profile of each cam segment in the Define Cam Profile dialog box.

To define the cam profile:

1. Select the cam, and click **Define Cam Profile** on the Shape tab of the Object Info palette.

   The Enter Value dialog box opens.

2. Enter the number of cam segments and click **OK**.

   The Define Cam Profile dialog box opens for specifying the parameters for each segment. Click **OK** to update cam diagram.

Click to show/hide the parameters.

Parameter	Description
Type of Motion	Select the type of motion for each segment
End Angle	Specify the end angle for the segment
Displacement	Specify the displacement at the end point of the segment

Animating Cam Movement

The effect of parameter changes to the cam design can be evaluated by animating the cam.

To animate the cam:

1. Select the cam, and in the Object Info palette, click **Animate**.

   The Rotate Cam dialog box opens.

2. Adjust the relative speed by moving the slider along the Speed bar.
3. Click **Rotate**.

   The cam rotates at the relative speed specified and the follower moves accordingly.

   Since the animation is for illustrative purposes only, the **Cam Speed** parameter does not affect the animation. However, the animation is affected by the selected speed in the Rotate Cam dialog box; the larger the increment, the faster the cam appears to rotate. In addition, the computer processor speed also affects the apparent animation speed.

4. To stop the cam rotation, press any key or click anywhere on the screen.

Resetting the Cam to Zero Degrees

Once the cam rotation stops, the position of the cam remains where it was when it was stopped. To reset the cam to zero degrees (original position), click **Reset Cam to 0** on the Shape tab of the Object Info palette.

Cam Data Worksheets

Two worksheets are defined in the Cam (Imperial).sta and Cam (Metric).sta templates:

- **Cam Data** – lists cam data, such as rotation angle, follower displacement, velocity, and acceleration
- **Cam Properties** – lists the cam parameters from the Object Info palette

The worksheets are accessed by selecting them from the **Window > Worksheets** menu or from the Resource Browser. The Cam Properties worksheet updates when the Define Cam Profile dialog box is exited (see “Defining the Cam

Profile" on page 1878). The Cam Data worksheet updates when the cam properties are changed. The worksheets can be placed on the drawing as graphic objects, if desired (see "Worksheets as Graphic Objects" on page 1360).

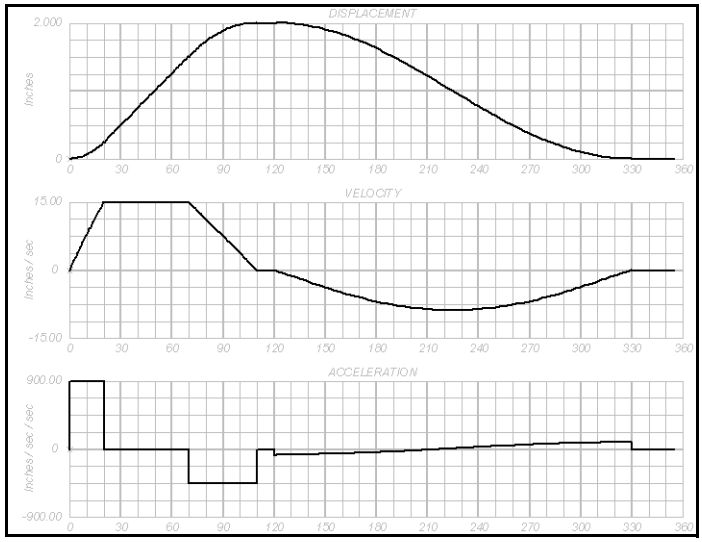
To prevent slower performance, when changing cam properties, keep the Cam Data worksheet closed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cam Template
Cam Properties
Cam Diagrams

Cam Diagrams

Diagrams of the cam follower displacement, velocity and acceleration are created automatically from the current cam properties.



The diagrams exist on the Diagrams layer.

To display or update the diagrams:

1. Select the **Draw Cam Diagrams** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Draw Cam Diagrams**
   - Landmark workspace: **Landmark > Machine Design > Draw Cam Diagrams**
   - Spotlight workspace: **Spotlight > Machine Design > Draw Cam Diagrams**
2. The cam diagrams are displayed. To return to the cam, select **Cam** from the Layers list on the View bar.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Cam Template
Cam Properties
Cam Data Worksheets

Geneva Mechanism

The Geneva mechanism device produces intermittent rotational motion of the driven part (index plate) while the driver rotates at a constant speed.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Geneva Mechanism Template
Geneva Mechanism Properties
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Animating Geneva Mechanism Movement

Geneva Mechanism Template

The Geneva mechanism is available in the Geneva Mechanism (Imperial).sta and Geneva Mechanism (Metric).sta template files.

To open the Geneva mechanism template:

1. Select **File > New**.

   The Create Document dialog box opens.

2. Select **Use document template**, and choose the Geneva Mechanism (Imperial).sta or Geneva Mechanism (Metric).sta template.

3. Click **OK**.

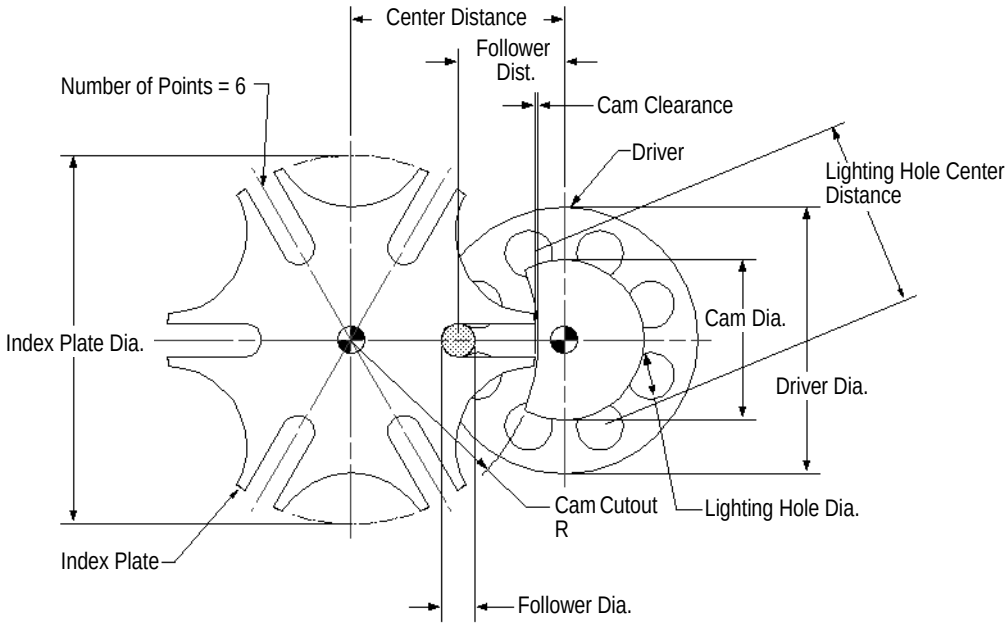
   The new file opens with a Geneva mechanism inserted by default.

   The recommended procedure to include a Geneva mechanism in a Vectorworks drawing is to copy and paste it into the drawing and convert it to a group by selecting it and selecting **Modify > Convert > Convert to Group**.

Geneva Mechanism Properties
Animating Geneva Mechanism Movement

Geneva Mechanism Properties

The Geneva mechanism's properties can be edited in the Object Info palette.



Click to show/hide the parameters.

Parameter	Description
Number of Points	Enter the number of points on the index plate
Center Distance	Enter the distance between the center of the index plate and the driver

Parameter	Description
Follower Dia.	Enter the follower diameter
Cam Clearance	Enter the cam clearance
Cam Dia.	Enter the cam diameter
Driver Dia.	Enter the driver diameter
Index Plate Dia. (Ref.)	Displays the diameter of the index plate
Follower Dist. (Ref.)	Displays the follower distance
Cam Cutout Radius (Ref.)	Displays the cam cutout radius
Lighting Holes	Select to place lighting holes on the driver
Number	Enter the number of lighting holes
Diameter	Enter the diameter of the lighting holes
Center Distance	Enter the distance between the center of the lighting holes about the driver
Show Center Marks	Select to place center mark symbols on the index plate and cam/driver
Show Center Lines	Select to draw the index plate and cam/driver with center lines

Geneva Mechanism Template
Animating Geneva Mechanism Movement

Animating Geneva Mechanism Movement

The effect of parameter changes to the Geneva mechanism design can be evaluated by animating the Geneva mechanism.

To animate the Geneva mechanism:

1. Select the Geneva mechanism, and in the Object Info palette, click **Animate**.

   The Rotate Geneva Mechanism dialog box opens.
2. Adjust the relative speed by moving the slider along the Speed bar.
3. Click **Rotate**.

   The Geneva mechanism rotates at the relative speed specified.
4. To stop the Geneva mechanism rotation, press any key or click anywhere on the screen.

Resetting the Geneva Mechanism to Zero Degrees

Once the mechanism rotation stops, the position of the mechanism remains where it was when it was stopped. To reset the mechanism to zero degrees (original position), click **Reset Mechanism to 0** on the Shape tab of the Object Info palette.

Notes Manager Database Format

The Notes Manager, in version 11 and later, can use a database from an outside source, provided that it has been formatted appropriately. Data files must be in one of two formats: a version 11 or later format, or the format used by the Notes Manager in versions of Vectorworks software previous to version 11. The format used by previous versions may be somewhat simpler to use, and is automatically converted to the current format by the Notes Manager.

Formatting by Previous Version

The database format must adhere to the following requirements to be recognized as a database from versions of the Notes Manager prior to 11.

- The first line must contain only a numeric value
- The rest of the file must contain notes with the following data separated by tabs:
  - a numeric value
  - a dash
  - the section name
  - the description
  - the note text
- Additional tabs or carriage returns are not allowed within each field
- There can be no blank lines before the end of the file

This format is illustrated by the following example.

```
3
00	-	Section 1	Description 1	This is keynote 1.
00	-	Section 1	Description 2	This is keynote 2.
00	-	Section 1	Description 3	This is keynote 3.
```

Formatting by Current Version

The database format must adhere to the following requirements to be recognized as a database from version 11 and later Notes Manager.

- The first line must contain the word “NotesManager11DataFile”
- The second line must contain the database ID (unique for all databases used)
- The rest of the file must contain notes with the following data separated by tabs:
  - a note ID (unique for all notes in the file)
  - the section name
  - the description
  - the note text
- After the notes, an optional section can identify which description was last selected in each section
- Additional tabs or carriage returns are not allowed within each field
- There can be no blank lines before the end of the file

The database and note IDs are in the format of “ID_#”, where “#” represents any numeric value with not more than 14 digits and no decimal point. Notes Manager does not change these IDs after creation, but when new notes are added, it uses IDs similar to those shown in the following example.

```
Notes Manager11DataFile
ID_04030110200020	General Project Notes	Scope of Work	Project located at ‘ADDRESS’,’CITY’,’STATE’, county of ‘COUNTY’.
ID_04030110200021	General Project Notes	Accessibility Compliance	Project complies with accessibility standards.
```

ID_04030110200022	General Project Notes	Involved Parties	"Owner" refers to 'OWNER ID' or its authorized representatives.
0	End of Notes		
General Project Notes	Scope of Work		

D Adding User-defined Information to Commands

Certain commands permit adding to the list of available values for certain parameters (for example, the **Material** list for the Spring Calculator or the **Units** list for the **Simple Beam** command).

Parameters Allowing Addition of User-defined Values in the GetUserData.vss File			
Group	Plug-in Object	Parameter	Dialog Box
1	Spring Calculator	Units (inches, mm and cm only)	Spring Calculator
	Shaft Analysis	Outside Diameter	Shaft Analysis
	Simple Beam	Beam Length	Simple Beam: Set Beam Properties
	Simple Beam	Deflection Units	Simple Beam: Set Beam Properties
2	Simple Beam	These items are the singular form of the names in Group 1 and are used when writing the values to a worksheet and a text block on the drawing	Not applicable
3	Simple Beam	Load Units	Simple Beam: Add/Remove/ Change Loads
4	Simple Beam	These items are the singular form of the names in Group 3 and are used when writing the values to a worksheet and a text block on the drawing	Not applicable
5	Simple Beam	Section Modulus Units	Simple Beam: Set Beam Properties
6	Simple Beam	Moment of Inertia Units	Simple Beam: Set Beam Properties
7	Shaft Analysis	Shear Modulus Units	Shaft Analysis
	Simple Beam	Modulus of Elasticity Units	Simple Beam: Set Beam Properties
8	Spring Calculator	Material	Spring Calculator
9	Shaft Analysis	Twisting Moment Units	Shaft Analysis

Adding User-defined Spring Calculator Materials
Adding User-defined Simple Beam Units

D Adding User-defined Spring Calculator Materials

The following example shows how to add to the list of materials available to the Spring Calculator.

To add to the list of Spring Calculator materials:

1. Make a backup copy of the GetUserData.vss file, located in Plug-Ins\VW_Mech\Includes. Keep the backup copy in case there is a need to restore the GetUserData.vss file to its original state.
2. Open the GetUserData.vss file in any word processor and locate group 8 in the getUserString function.
3. Using the next available sequential number, add the new **Material** to the end of the list (in this example, Beryllium Copper). Enclose the name in single quotes and put a semi-colon at the end of the line. The new line should read:

   16: getUserString := ‘Beryllium Copper’;

```
{* Spring Calculator - Material *}
8: BEGIN
        CASE item OF
                1: getUserString := '<Other>';
                2: getUserString := 'Music Wire - Up to 0.032" (0.81mm)';
                3: getUserString := 'Music Wire - 0.033" to 0.063" (0.82mm to 1.60mm)';
                4: getUserString := 'Music Wire - 0.064" to 0.125" (1.61mm to 3.18mm)';
                5: getUserString := 'Music Wire - 0.126" to 0.250" (3.19mm to 6.35mm)';
                6: getUserString := 'Hard Drawn MB - Up to 0.032" (0.81mm)';
                7: getUserString := 'Hard Drawn MB - 0.033" to 0.063" (0.82mm to 1.60mm)';
                8: getUserString := 'Hard Drawn MB - 0.064" to 0.125" (1.61mm to 3.18mm)';
                9: getUserString := 'Hard Drawn MB - 0.126" to 0.250" (3.19mm to 6.35mm)';
                10: getUserString := 'Oil Tempered MB';
                11: getUserString := 'Stainless Steel - Types  302, 304, 316';
                12: getUserString := 'Stainless Steel - Type 17-7 PH';
                13: getUserString := 'Stainless Steel - Type 420';
                14: getUserString := 'Stainless Steel - Type 431';
                15: getUserString := 'Spring Brass';
                16: getUserString := 'Beryllium Copper';
        END;    {of CASE item}
END;    {of group = 8}
```

4. Locate group 8 in the getUserData function.
5. Using the next available sequential number, add the new **Mod. of Elasticity** value (in this example, pounds per square inch). The sequential number added here must correspond to the number added in the getUserString function. Put a semi-colon at the end of the line; do not use single quotes or commas in the value. The new line should read:

   16: getUserData := 7000000;

6. Change the value of item 0 to the new group number, which indicates the number of available values (in this example, 16).

```
{* Spring Calculator - Material *}
{* Units = pounds per square inch           *}
8: BEGIN
        CASE item OF
                0: getUserData := 16;
                1: getUserData := 0;
                2: getUserData := 12000000;
                3: getUserData := 11850000;
                4: getUserData := 11750000;
                5: getUserData := 11600000;
                6: getUserData := 11700000;
                7: getUserData := 11600000;
                8: getUserData := 11500000;
                9: getUserData := 11400000;
                10: getUserData := 11200000;
                11: getUserData := 10000000;
                12: getUserData := 10500000;
                13: getUserData := 11000000;
                14: getUserData := 11400000;
                15: getUserData := 5000000;
                16: getUserData := 7000000;
        END;    {of CASE item}
END;    {of group = 8}
```

7. Save the file. The next time the Spring Calculator is run, the new item is available in the Material list, and when selected, the new value is available in the Mod. of Elasticity field. When executed, the Spring Calculator automatically recalculates the value to the user-specified units. (For beryllium copper, the value is 7,000,000 lb/in^2.)

If the Spring Calculator was used during the current session, the Vectorworks program needs to be restarted for the changes to take place.

D Adding User-defined Simple Beam Units

The following example shows how to add to the list of available units for the **Simple Beam** command.

To add to the list of units for the Simple Beam command:

1. Make a backup copy of the GetUserData.vss file located in Plug-Ins\VW_Mech\Includes. Keep the backup copy in case there is a need to restore the GetUserData.vss file to its original state.
2. Open the GetUserData.vss file in any word processor and locate group 1 in the getUserString function.
3. Using the next available sequential number, add the new **Units** to the end of the list (in this example, Yards). Enclose the name in single quotes and put a semi-colon at the end of the line. The new line should read:

   6: getUserString := ‘Yards’;
4. Locate group 2 in the getUserString function.
5. Using the next available sequential number, add the new **Units** to the end of the list. (In this example, type yard in the singular form; this value is used in certain output for the **Simple Beam** command and must have an entry corresponding to the same item in group 1).

   Enclose the name in single quotes and put a semi-colon at the end of the line. The new line should read:

   6: getUserString := ‘Yard’;

```
FUNCTION getUserString (group, item : INTEGER) : STRING;
BEGIN
	CASE group OF
		{* Spring Calculator - Units (items 1, 3 and 4 only  *)
		{* Shaft Analysis - Outside Diameter                 *)
		{* Simple Beam: Beam Properties - Beam Length        *)
		{* Simple Beam: Beam Properties - Deflection Units   *)
		1: BEGIN
			CASE item OF
				1: getUserString := 'Inches';
				2: getUserString := 'Feet';
				3: getUserString := 'mm';
				4: getUserString := 'cm';
				5: getUserString := 'Meters';
				6: getUserString := 'Yards';
			END;   {of CASE item}
		END;   {of group = 1}

		{* Simple Beam *)
		{* Note: These values are the singular form of the values in group 1 and are only used
for output. There are no data units associated with this group *}
		2: BEGIN
			CASE item OF
				1: getUserString := 'Inch';
				2: getUserString := 'Foot';
				3: getUserString := 'mm';
				4: getUserString := 'cm';
				5: getUserString := 'Meter';
				6: getUserString := 'Yard';
			END;   {of CASE item}
		END;   {of group = 2}
```

6. Locate group 1 in the getUserData function.
7. Using the next available sequential number, add the new **Units** to the end of the list (in this example, units per inch of a yard: 1/36). The sequential number added here must correspond to the number added in the getUserString function. Put a semi-colon at the end of the line; do not enclose the name in single quotes. The new line should read:

   6: getUserData := 1/36;
8. Change the value of item 0 to the new group number, which indicates the number of available values (in this example, 6).

```
FUNCTION getUserData (group, item : INTEGER) : REAL;
BEGIN
      CASE group OF
            {* Spring Calculator - Units (items 1, 3 and 4 only   *)
            {* Shaft Analysis - Outside Diameter                  *)
            {* Simple Beam: Beam Properties - Beam Length         *)
            {* Simple Beam: Beam Properties - Deflection Units    *)
            {* Units = units per inch                             *)
            1: BEGIN
                  CASE item OF
                        0: getUserData := 6;
                        1: getUserData := 1;
                        2: getUserData := 1/12;
                        3: getUserData:= 25.4;
                        4: getUserData:= 2.54;
                        5: getUserData:= 0.0254;
                        6: getUserData:= 1/36;
                  END;   {of CASE item}
            END;   {of group = 1}
```

9. Save the file. The next time the **Simple Beam** command is used, the new item is available in the Beam Properties dialog box.

   This change also affects the Shaft properties of the **Shaft Analysis** command. These values are also used by the Spring Calculator Units, but only items 1, 3, and 4 are applicable, so the new units (Yards) will not show up in this field.

Simple Beam and Spring Calculator Examples

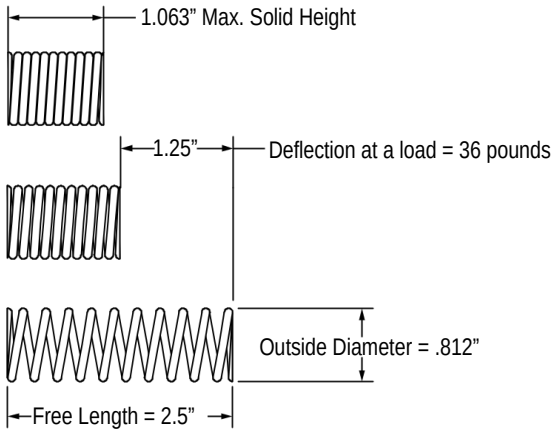
This section provides examples of using the **Spring Calculator** and **Simple Beam** commands to accomplish specific tasks.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Spring Calculator
Simple Beam

Spring Calculator

The spring calculator can determine spring rates and unit stresses of round wire helical coil compression springs with known parameters. It can also be used to design a spring knowing the working values. This example is based on the following compression spring with closed and ground ends of music wire.

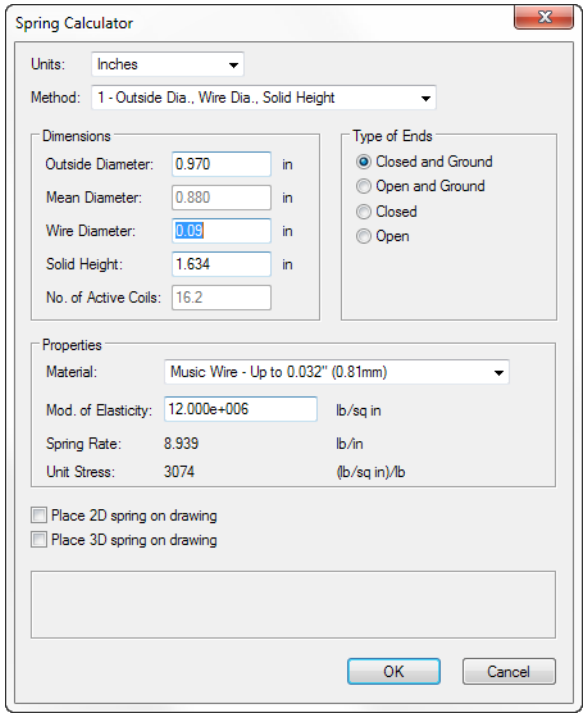


To calculate spring rate and unit stress:

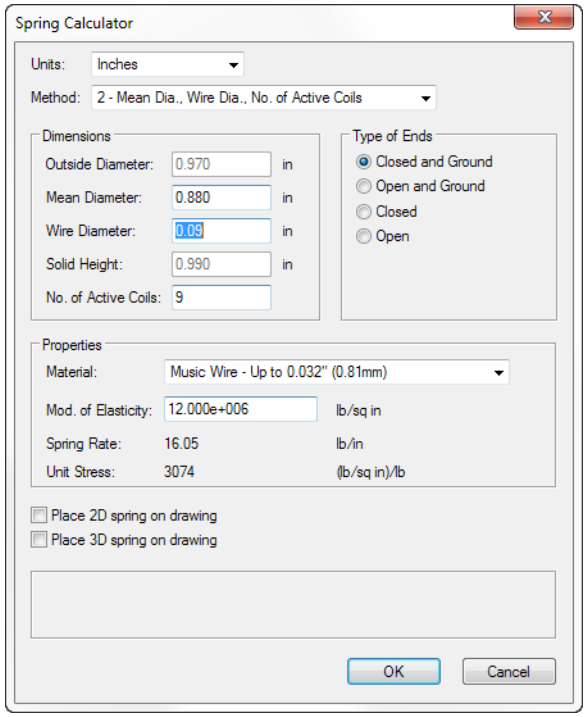
1. Determine the required spring rate based on the deflection at a load divided by the difference in the working length and the free length. For this example, the desired spring rate equals 28.8 lb/in (36 lb/1.25 in = 28.8 lb/in).
2. Select the **Spring Calculator** command from the appropriate menu:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Architect workspace: **AEC > Machine Design > Spring Calculator**
- Landmark workspace: **Landmark > Machine Design > Spring Calculator**
- Spotlight workspace: **Spotlight > Machine Design > Spring Calculator**

The Spring Calculator dialog box opens.



3. As shown in the above dialog box, select **1 - Outside Dia., Wire Dia., Solid Height** from the **Method** list. After entering the known values, calculate a spring rate close to the desired value by trying several standard wire diameter values. Adjust the material in the **Material** list to fit the wire diameter used. Here, a wire diameter of .090” gives a spring rate of 8.94 lb/in.
4. In the **Method** list, select **2 - Mean Dia., Wire Dia., No. of Active Coils**.



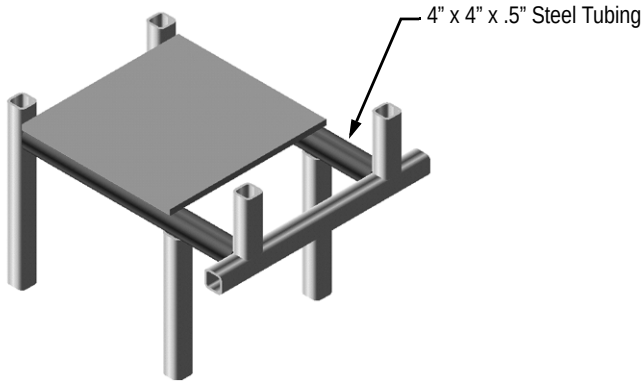
5. As shown in this dialog box, vary the wire diameter and number of active coils to get a spring rate close to the required spring rate. A wire diameter of .095” and 11 active coils gives a spring rate of 16.3 lb/in, but the solid height is 1.24”, which is too high. A wire diameter of .090” and 9 active coils, however, gives a spring rate of 16.1 lb/in, and a solid height of .990”, which is within acceptable limits.
6. Finally, check the stresses applied to the spring to verify that they are within acceptable limits. With a unit stress of 3074 (lb/sq in)/lb, multiply by 36 to obtain 110,663 lb/sq in. With a solid height of .990”, the stress will be:

   (2.500-0.900)in x 16.1 lb/in x 3074 (lb/sq in)/lb = 79,200 lb/sq in
7. This value is below the safe working stress of 111,000 lb/sq in for this material and wire size.

D Simple Beam

This example shows how the Simple Beam commands can be used to find the stresses on the horizontal elements of a certain machine part.

Data:
Length = 48”
Distance between supports = 36”
Distributed load = 600lb/in
Concentrated load = 2000lb
Tubing = 4” x 4” x 1/2” Steel
Section Modulus = 6.12in^3
Moment of Inertia = 12.26in^4
Modulus of Elasticity = 29E+6lbsq in



To calculate the stresses:

1. Select **File > New**.

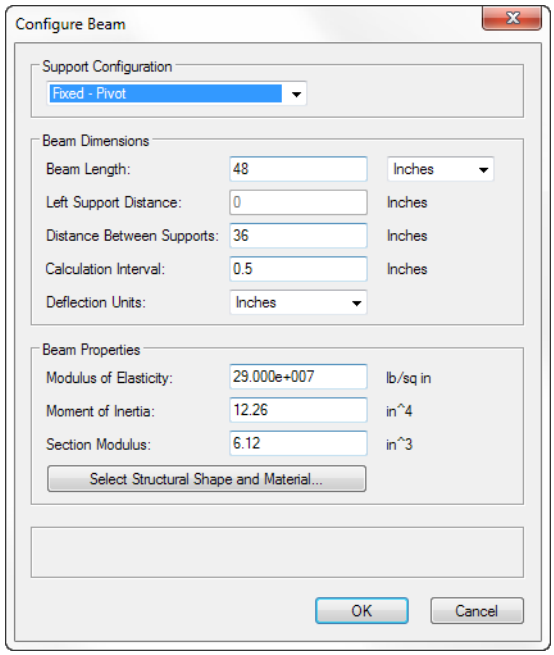
The Create Document dialog box opens.

2. Select **Use document template**, and choose the Simple Beam (Imperial) or Simple Beam (Metric).sta template.
3. Click **OK**.
4. Select the **Simple Beam** command from the appropriate menu:
   - Architect workspace: **AEC > Machine Design > Simple Beam**
   - Landmark workspace: **Landmark > Machine Design > Simple Beam**
   - Spotlight workspace: **Spotlight > Machine Design > Simple Beam**

   The Simple Beam dialog box opens.
5. Click **Configure Beam**.

   The Configure Beam dialog box opens.
6. Specify the beam properties. In this example, enter the following values:

Configuration: Fixed - Pivot
Length: 48
Units: Inches
Distance Between Supports: 36
Section Modulus: 6.12 in^3
Moment of Inertia: 12.26 in^4
Modulus of Elasticity: 29E+6 lb/sq in
Calculation Interval: 0.5
Deflection Units: Inches

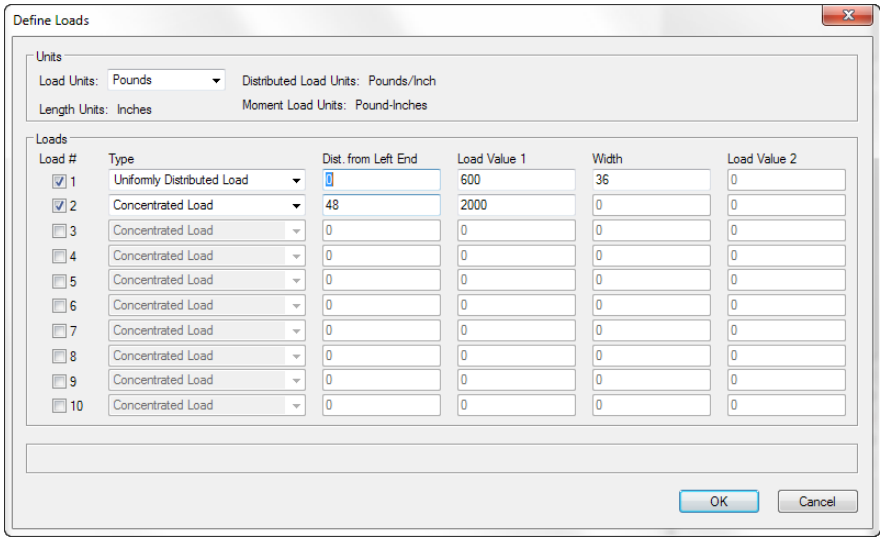


To obtain the section modulus and moment of inertia of the tubing, select a tubing shape from the Resource Browser and place an instance of it on the drawing. Convert the tubing to a group and then use the **Engineering Properties** command to obtain the properties. For more information on the **Engineering Properties** command, see “Obtaining Engineering Properties” on page 1821.

7. Click **OK** to close the Beam Properties dialog box.
8. Click **Define Loads** on the Simple Beam dialog box.

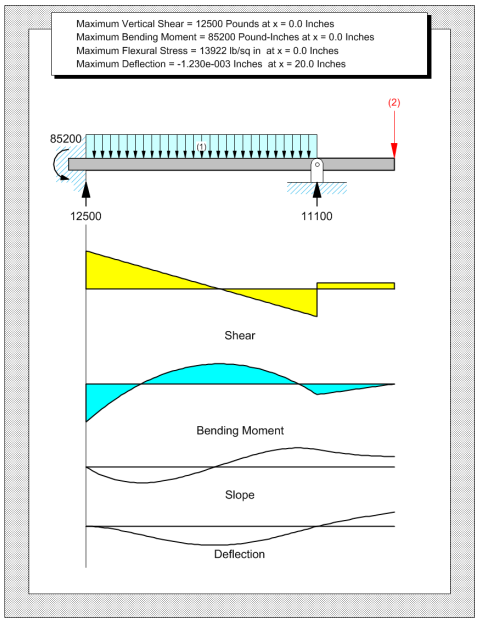
   The Define Loads dialog box opens.
9. Specify the load properties. In this example, use the following values:

Load Units: Pounds
Load #1
Type: Uniformly Distributed
Distance from Left End: 0
Load Value 1: 600
Width: 36
Load #2
Type: Concentrated Load
Distance from the Left End: 48
Load Value 1: 2000



10. Click **OK** close the Define Loads dialog box.
11. In the Simple Beam dialog box, click **Calculate Reactions**, and then click **Calculate Stresses and Deflection**, and then click **OK**.

    A simple beam diagram is created based on the information provided. See “Simple Beam” on page 1888 for more information on what is depicted in the diagram.



A L Layer, Class, and Viewport Standards

The standards for the Vectorworks Architect and Landmark products are centered around the concept of viewports on sheet layers. A project file contains a number of viewports; each viewport consists of a particular combination of visible layers and classes.

This section provides information about the viewport, layer, and class standards in Vectorworks Architect and Landmark products. Modifying these standards is not recommended, as it requires a large effort to manage and coordinate standards information.

Using the Layermap Worksheet
The Layermap Worksheet
Viewport/View Types
Standard Layer Visibility in Viewports/Views
Design Layer Types in the Standards
Class Visibility in the Standards

Using the Layermap Worksheet

The creation of layers, classes, and viewports or views by the setup commands (**Create Standard Viewports** in Vectorworks Architect and **Standard Naming** in Vectorworks Architect and Landmark) is controlled through the use of the worksheet LayerMap.G, in the **VA Setup Data-Imperial.vwx** or **VA Setup Data-Metric.vwx** file ([Vectorworks]\Plug-Ins\Common\Data). This worksheet contains the predefined class, layer, attribute, and visibility information needed to generate a completely ready-to-use project file at any scale or level of complexity.

The LayerMap.G worksheet applies to viewports on sheet layers created with the **Create Standard Viewports** command, or views created with the **Create Corresponding View for Each Viewport** option in the Create Standard Viewports dialog box.

The settings in the LayerMap.G worksheet are automatically applied when a new file is opened and one of the Setup commands is selected. If the LayerMap.G worksheet contained in the **VA Setup Data-Imperial.vwx** or **VA Setup Data-Metric.vwx** file is edited, all sheets created on that machine will use the new standards. However, a copy of the LayerMap.G worksheet can be included in a file (by importing it) and edited there. These custom settings will supersede any automated settings for that file only. This is a convenient way of sharing standards between offices and users, without modifying the default standards on an individual machine.

If a Vectorworks file contains a LayerMap.G worksheet, when the **Create Standard Viewports** command is selected, the Import LayerMap.G dialog box opens. Specify which worksheet to use.

Click to show/hide the parameters.

Parameter	Description
Use the existing LayerMap.G	Uses the LayerMap.G worksheet present in the file rather than the standard worksheet
Import and use the standard LayerMap.G	Imports and applies the standard LayerMap.G worksheet (from the VA Setup Data.vwx file) rather than using the settings from the existing imported worksheet
Delete the LayerMap.G worksheet when finished	Deletes the LayerMap.G worksheet from the file; this is recommended when using the standard worksheet

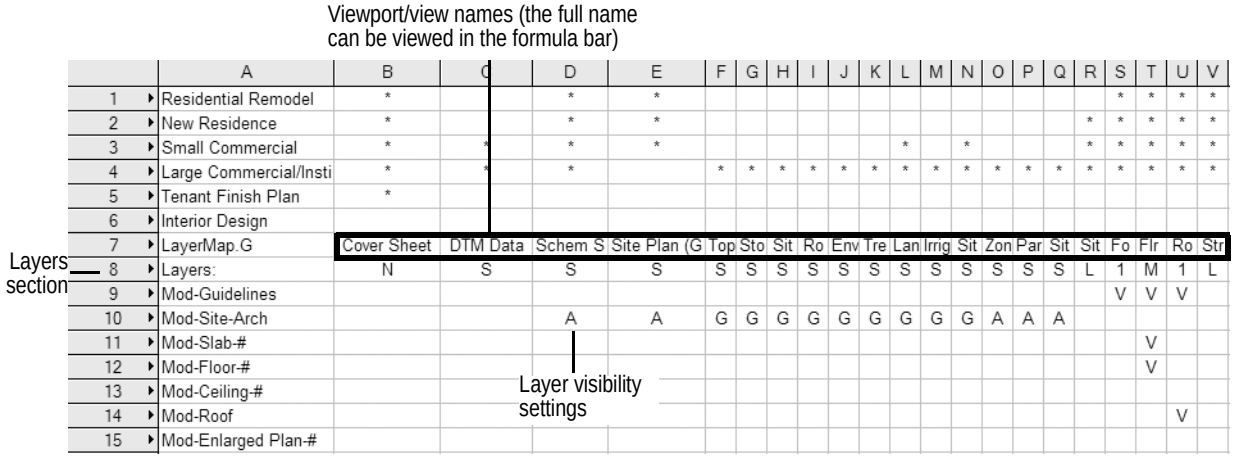
The Layermap Worksheet

The Layermap Worksheet

The Layermap worksheet, located in the **VA Setup Data-Imperial.vwx** or **VA Setup Data-Metric.vwx** file, begins with a listing of project types in the first six rows (before the LayerMap.G row). These project types are used by the legacy command **Create Standard Views**, and are not used by the **Create Standard Viewports** command. As such, they are no longer supported.

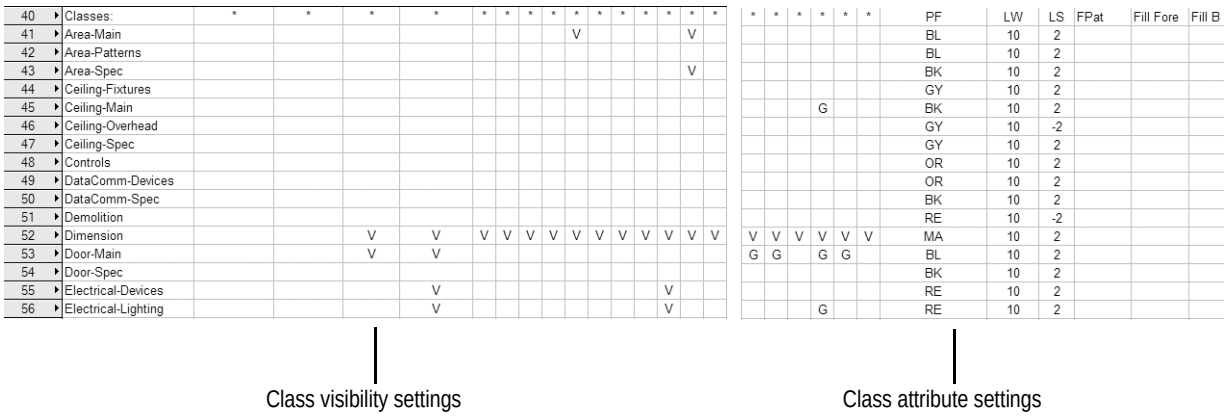
Following the legacy project types are the names of the viewports or views used by the **Create Standard Viewports** command. Below the name is the viewport/view type, which controls under which drawing type the viewport/view appears in the Create Standard Viewports dialog box.

After the project types section is the layers section. This section of the Layermap worksheet controls the initialization and setup of layers in a specific viewport/view. Based on the information in this section, the **Create Standard Viewports** command generates a complete array of layers with the proper settings to correctly display a viewport or view.



	A	B	C	D	E	F	G	H	I	J	K	L	M	N	O	P	Q	R	S	T	U	V
1	Residential Remodel	*		*	*														*	*	*	*
2	New Residence	*		*	*													*	*	*	*	*
3	Small Commercial	*	*	*	*							*		*				*	*	*	*	*
4	Large Commercial/Insti	*	*	*		*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*
5	Tenant Finish Plan	*																				
6	Interior Design																					
7	LayerMap.G	Cover Sheet	DTM Data	Schem S	Site Plan (G	Top	Sto	Sit	Ro	Env	Tre	Lan	Irrig	Sit	Zon	Par	Sit	Sit	Fo	Flr	Ro	Str
8	Layers:	N	S	S	S	S	S	S	S	S	S	S	S	S	S	S	S	L	1	M	1	L
9	Mod-Guidelines																		V	V	V	
10	Mod-Site-Arch			A	A	G	G	G	G	G	G	G	G	G	A	A	A					
11	Mod-Slab-#																			V		
12	Mod-Floor-#																			V		
13	Mod-Ceiling-#																					
14	Mod-Roof																				V	
15	Mod-Enlarged Plan-#																					

The final section in the Layermap worksheet is the classes section. This section controls class setup and initialization for each viewport/view. As with the layer section, the Setup commands use the information to generate a full class setup configuration for each viewport/view in a project file.



	Classes:	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	*	PF	LW	LS	FPat	Fill Fore	Fill B
41	Area-Main								V				V								BL	10	2			
42	Area-Patterns																				BL	10	2			
43	Area-Spec												V								BK	10	2			
44	Ceiling-Fixtures																				GY	10	2			
45	Ceiling-Main																			G	BK	10	2			
46	Ceiling-Overhead																				GY	10	-2			
47	Ceiling-Spec																				GY	10	2			
48	Controls																				OR	10	2			
49	DataComm-Devices																				OR	10	2			
50	DataComm-Spec																				BK	10	2			
51	Demolition																				RE	10	-2			
52	Dimension			V	V	V	V	V	V	V	V	V	V	V	V	V	V	V	V	V	MA	10	2			
53	Door-Main			V	V										G	G		G	G		BL	10	2			
54	Door-Spec																				BK	10	2			
55	Electrical-Devices				V								V								RE	10	2			
56	Electrical-Lighting				V								V							G	RE	10	2			

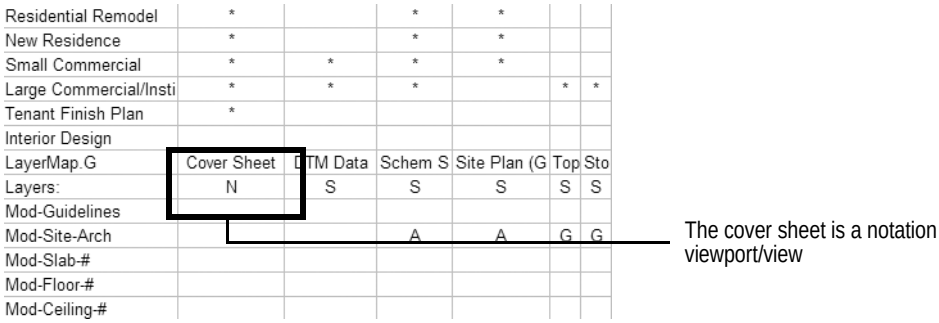
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Using the Layermap Worksheet
Viewport/View Types
Standard Layer Visibility in Viewports/Views
Design Layer Types in the Standards
Class Visibility in the Standards

A Viewport/View Types

Viewport/view types are found directly under the viewport/view names in the Layermap worksheet. The identifier directly beneath the viewport/view name indicates its drawing type. Viewport/view types control how the viewport/view is generated, and are used by the **Create Standard Viewports** command to properly configure the project document. The viewport/view type identifier may take on one of the following values:
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Identifier	Drawing Type
1	Project plan view/viewport (one viewport/view only)
L	Auxiliary viewport/view (create a user-specified number of viewports/views)
M	Floor plan viewport/view (create one viewport/view for each floor in the project)
N	Notation viewport/view (create a user-specified number of viewports/views)
S	Site plan viewport/view (create one viewport/view)

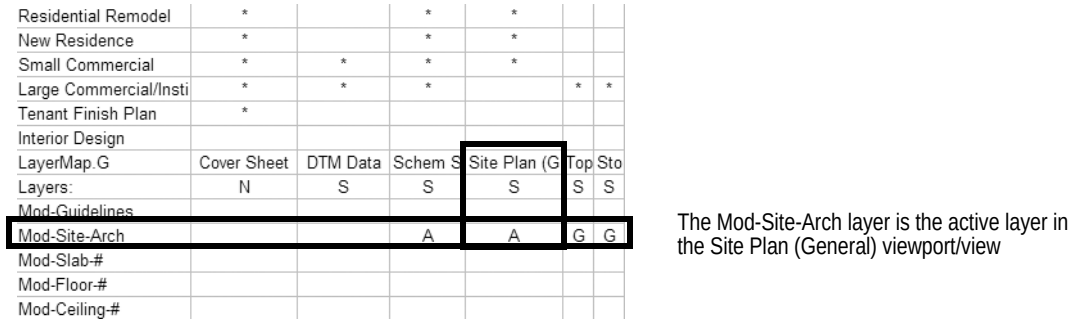


A Standard Layer Visibility in Viewports/Views

In the layers section of Layermap worksheet, layer names are listed down the left side of the worksheet in the first column. This listing represents the available layers that can be included in a viewport/view. Layer names that are used when generating multiple layers (for example, for the floor viewports/views of a multi-floor building) are indicated by a pound sign (#) suffix.

The visibility status of a specific layer is indicated by an alphabetic identifier located in the worksheet cell which cross-references the layer and the viewport/view in which it will be a component. The identifier can take on one of the following values:

Identifier	Meaning
A	Active layer
V	Visible
G	Grayed
I	Invisible
<no value>	Not created for viewport/view; set to invisible if existing

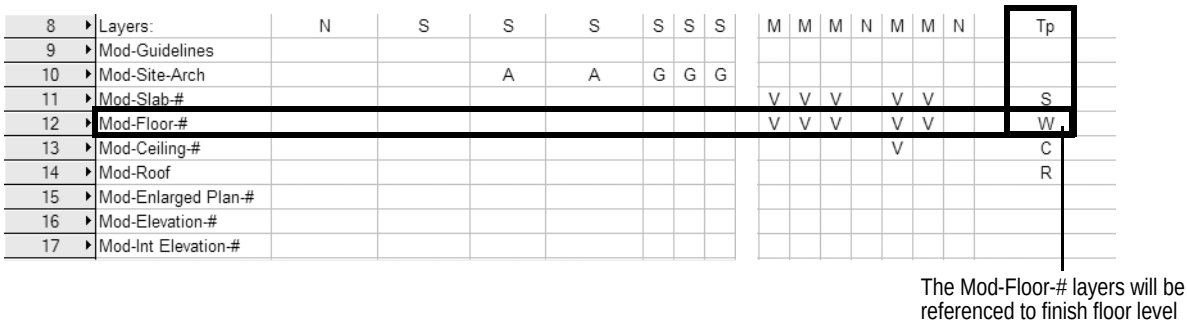


Design Layer Types in the Standards

The Layermap worksheet contains a specialized set of identifiers which control how model design layers are configured for a project file. These identifiers, which are located on the extreme right of the Layermap worksheet opposite the model layers, are used by the Setup commands to correctly configure a set of model layers for the project file.

Model layer type identifiers can take on one of the following values:

Identifier	Meaning
C	Layer w/ ceiling referenced information (one per floor)
F	Foundation layer (one per project)
R	Roof layer (one per project)
S	Slab layer (one per floor)
W	Layer w/ floor referenced information (one per floor)
<no value>	Supplemental model layer



Class Visibility in the Standards

In the class section of the Layermap worksheet, class names available for inclusion in the viewports/views are listed down the left side of the worksheet in the first column. The visibility of a specific class is indicated by an alphabetic identifier, which is located in the worksheet cell which cross-references the layer and the viewport/view.

The identifier can take on one of the following values:

Identifier	Meaning
V	Visible
G	Grayed
I	Invisible
<no value>	Not created for viewport/view; set to invisible if existing

An explanation of the specific uses of and applications for classes, including AIA layer name equivalents, is provided in PDF format (VA Sheet-Layer-Class stds.pdf) included with the Vectorworks Architect product and available in [Vectorworks]\Extras.

Default class attributes, such as line style and pen color, are no longer controlled from the Layermap worksheet.

AL Auto-classing Objects

Certain plug-in objects and symbols have specific default classes, which coordinate with the Layermap worksheet class name standards. The plug-in objects listed here are automatically classed when they are inserted into a file that is set up to use auto-classing (when **Enable Auto-classing** is selected in the Standard Naming dialog box). If auto-classing is not enabled, these objects are set to a default class when they are inserted into a file.

Object	Default Class	Vectorworks Product
Base Cabinet	Millwork-Main	Architect
Bath-Shower	Plumbing-Fixtures	Architect
Collar	Structural-Framing	Architect
Column	Structural-Columns	Architect, Landmark
Comm Device	DataComm-Devices	Architect
Compartment Sink	Plumbing-Fixtures	Architect
Counter Top	Millwork-Main	Architect
Desk	Furniture-Main	Architect
Door	Door-Main	Architect, Landmark
Drilled Footing	Structural-Footings	Architect
Drip Emitter	Irrigation-SprayPat	Landmark
Escalator	Vert Trans-Main	Architect
Fireplace	Fixtures-Main	Architect
Grab Bars	Fixtures-Main	Architect
Grade	Site_Util_Storm	Landmark
Guardrail (Curved)	Site-Improvements	Architect, Landmark
Guardrail (Straight)	Site-Improvements	Architect, Landmark
Handrail (Curved)	Fixtures-Main	Architect, Landmark
Handrail (Straight)	Fixtures-Main	Architect, Landmark
Hardscape (tag)	Site-Hardscape Comp-Spec	Landmark
Hardscape (border)	Site-Hardscape Comp-Border Joint	Landmark
Hardscape (main area)	Site-Hardscape Comp-Main Joint	Landmark
Hip and valley rafters	Structural-Framing	Architect
HVAC Damper	HVAC-Duct/Equipment	Architect
HVAC Diffuser	HVAC-Diffusers	Architect
HVAC Elbow Duct	HVAC-Duct/Equipment	Architect
HVAC Flex Duct	HVAC-Duct/Equipment	Architect
HVAC Outlet	HVAC-Duct/Equipment	Architect
HVAC Splitter	HVAC-Duct/Equipment	Architect
HVAC Straight Duct	HVAC-Duct/Equipment	Architect

Object	Default Class	Vectorworks Product
HVAC Transition	HVAC-Duct/Equipment	Architect
HVAC Vertical Duct	HVAC-Duct/Equipment	Architect
HVAC Vertical Elbow	HVAC-Duct/Equipment	Architect
Irrigation Head	Irrigation-SprayPat	Landmark
Joist	Structural-Framing	Architect
Mullion	Wall-Ext-Glazed	Architect
North Arrow	Notes-Sheet	Architect, Landmark
Parking Spaces	Site-Paving-Marking	Architect, Landmark
Pilaster	Structural-Columns	Architect
Plate	Structural-Framing	Architect
Purlin	Structural-Framing	Architect
Rafter	Structural-Framing	Architect
Ramp	Vert Trans-Main	Architect, Landmark
Receptacle	Electrical-Devices	Architect
Reference Marker	Notes-Sheet	Architect, Landmark
Revision Marker	Notes-Sheet	Architect, Landmark
Ridge	Structural-Framing	Architect
Roadway (Poly)	• Site-Paving-Surface • Site-Paving-Curb • Site-Paving-Stations	Architect, Landmark
Roadway (Curved), Roadway (Custom Curb), Roadway (NURBS), Roadway (Straight), Roadway (Tee)	• Site-Paving-Surface • Site-Paving-Curb	Architect, Landmark
Shelving Unit	Furniture-Main	Architect
Simple Elevator	Vert Trans-Main	Architect
Site Modifiers with Grade Limits configuration	Site-DTM-Modifier	Architect, Landmark
Site Modifiers with Pad configuration	Site-DTM-Modifier	Landmark
Site Modifiers with Texture Bed configuration	Site-DTM-Modifier	Landmark
Space (2D) - boundary	Space-Main	Architect
Space (2D) - label	Space-Spec	Architect
Space (2D) - leader line	Space-Spec	Architect
Space (3D) - attributes	Space-Patterns	Architect
Stair	Vert Trans-Main	Architect, Landmark
Switch	Electrical-Devices	Architect
Table and Chairs	Furniture-Main	Architect, Landmark

Object	Default Class	Vectorworks Product
Toilet Stall	Fixtures-Main	Architect
Trimmer	Structural-Framing	Architect
Utility Cabinet	Millwork-Main	Architect
VBvisual Plant	Landscape-Plants	Renderworks
Wall Cabinet	Millwork-Main	Architect
Window	Window-Main	Architect, Landmark
Workstation Counter	Furniture-Main	Architect
Workstation Overhead	Furniture-Main	Architect
Workstation Panel	Furniture-Main	Architect
Workstation Pedestal	Furniture-Main	Architect

D Machine Design Class Standards

The Machine Design_Classes.sta file, located in the Standards folder, contains a number of pre-defined class standards.

To use the pre-defined classes in a drawing:

1. Select **Tools > Organization**.

   The Organization dialog box opens. Click the Classes tab.
2. Click **New**.

   The New Class dialog box displays.
3. Click **Import Classes** and choose Machine Design_Classes.sta from the list.
4. Select the desired classes and click **OK**.
5. Click **OK** in the Organization dialog box to return to the drawing.

   Click the classes list on the Data bar to list the classes that are available for use in the drawing.

A Project Preference Sets

The following table lists the files contained in a preference set folder, and the menu commands/tools/objects which use those preference files. A description of the information stored in each preference file is also provided.

Preference File	Used By	Information Stored
Door Hardware Library.txt	Door tool, Door Hardware Library	Door hardware set name and record values
Equipment Record.txt	VA Create Record	Format and default values of record
Equipment Schedule.txt	VA Create Schedule, VA Records and Schedules	Format of schedule
Plumbing Fixt Record.txt	VA Create Schedule	Format and default values of record

Preference File	Used By	Information Stored
Plumbing Schedule.txt	VA Create Schedule, VA Records and Schedules	Format of schedule
Room Finish Library.txt	Space tool, Room Finish Library	Room finish names and descriptions
VA2 Records and Schedules.txt	VA Records and Schedules	Format of records and schedules, and default values of records
VA_Project_Set.txt	Utility file for VA to ensure that the folder has not been moved	

Taken together, these files constitute a preference set; they must all be present in the folder of the "current preference set."

The Vectorworks Architect product ships with six predefined preference sets.

Name	Location
VA Defaults	[Vectorworks]\Plug-Ins\VW_Arch\Data\Prefs_Def
Prefs_01	[Vectorworks]\Plug-Ins\VW_Arch\Data\Prefs_01
Prefs_02	[Vectorworks]\Plug-Ins\VW_Arch\Data\Prefs_02
Prefs_03	[Vectorworks]\Plug-Ins\VW_Arch\Data\Prefs_03
Prefs_04	[Vectorworks]\Plug-Ins\VW_Arch\Data\Prefs_04
Prefs_05	[Vectorworks]\Plug-Ins\VW_Arch\Data\Prefs_05

Making **no** changes to the preferences in the Defaults folder is strongly recommended. System administrators should make this folder "read-only" (Windows) or "locked" (Mac) to prevent users from making changes. The remaining preference set folders (01 – 05) can be customized.

Unused folders can be removed, leaving only the one that will be used; new folders can be created. To create a new folder, simply copy one of the existing folders to a new location. Select **Tools > Options > VA Set Project Prefs** to specify the location of the new folder. The folder location is not restricted. In a workgroup environment, locating the folder on the network makes the same project preference set available to everyone.

~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

Working with Project Preference Sets

Reserved Names

There are a number of words that are reserved for specific use within the Vectorworks Landmark product. These words should not be used to name objects or symbols.

- 2D Contours to 3D Polys
- DTM Mod Record
- Elev Analysis
- ID Label
- LineLength
- None
- BADSHEET678
- Data
- Fit To Objects
- Irrigation Head
- Mod-Guidelines
- Plant
- Cut N Fill DTM
- Drawing Border
- Guides
- Landscape-Plant
- Mod-Site-Civil
- Plant Record
~~~~~~~~~~~~~~~~~~~~~~~~~~~~~~

- Polyline ID
- Project Plant List
- Project Symlst
- Raise/Lower Record
- Roadway (Curved)
- Roadway (Straight)
- Saved Sheets
- Setup Record
- Sheet Border
- Sheet-Common
- Site-DTM-Modifier
- Site-Improvement-Spec
- Slope Analysis
- Stake #
- Stake Object
- Stake Record
- Station Worksheet
- Survey Data
- Symbol List
- Symbol List Sym
- Symbol to Group
- VW_Land-Invisible
- Vegetation line
- ZVI Analysis

Vectorworks 2015 Keyboard Shortcuts

The Vectorworks program has keyboard and mouse shortcuts for many common tasks. Many (but not all) shortcuts can be customized; the headings below indicate whether the shortcuts can be modified, and where. Note that these tables list the default shortcuts for all workspaces; your workspace may not have all of the tool sets, tools, menus, and commands listed.

For your convenience, the shortcuts are available as a PDF file for printing from the help system Table of Contents.

Tool Shortcuts (Modify in Workspace Editor)

Basic Palette

Tool	Windows	Mac
Selection	X	X
Pan	H	H
Move Page	Alt+Z	Option+Z
Flyover	Shift+C	Shift+C
Zoom	C	C
Text	1	1
Callout	Alt+1	Option+1
2D Locus	0 (zero)	0 (zero)
Symbol Insertion	Alt+0 (zero)	Option+0 (zero)
Line	2	2
Double Line	Alt+2	Option+2
Rectangle	4	4
Rounded Rectangle	Alt+4	Option+4
Circle	6	6
Oval	Alt+6	Option+6
Arc	3	3
Quarter Arc	Alt+3	Option+3
Freehand	Alt+5	Option+5
Polyline	5	5
2D Polygon	8	8
Triangle	Shift+T	Shift+T
Double-line Polygon	Alt+8	Option+8
Regular Polygon	Alt+Shift+R	Option+Shift+R
Spiral	Alt+Shift+S	Option+Shift+S
Eyedropper	Shift+E	Shift+E
Visibility	V	V
Attribute Mapping	Shift+A	Option+A
Reshape	- (hyphen)	- (hyphen)
Rotate	Alt+=	Option+=
Mirror	=	=
Split	L	L
Connect/Combine	; (semi-colon)	Option+L
Trim	Alt+Shift+L	Option+Shift+L
Fillet	7	7
Chamfer	Alt+7	Option+7
Offset	Shift+- (hyphen)	Shift+- (hyphen)
Clip	Shift+N	Option+C

Basic Palette

Move by Points	Shift+M	Shift+M

Building Shell Tool Set

Tool	Windows	Mac
Wall	9	9
Round Wall	Alt+9	Option+9
Wall Join	Alt+J	Option+J
Component Wall Join	Alt+Shift+J	Option+Shift+J
Door	Alt+Shift+D	Option+Shift+D
Window	Shift+D	Option+Shift+W

3D Modeling Tool Set

Tool	Windows	Mac
Flyover	Shift+C	Shift+C
Set Working Plane	Shift+1	Shift+1
Align Plane	Alt+Shift+1	Option+Shift+1
Push/Pull	Shift+R	Shift+R
3D Locus	Shift+0 (zero)	Shift+0 (zero)
NURBS Curve	Shift+7	Shift+7
Sphere	Shift+3	Shift+3
Hemisphere	Alt+Shift+3	Option+Shift+3
Fillet Edge	Shift+F	Shift+F
Chamfer Edge	Shift+J	Shift+J
Shell Solid	Shift+G	Shift+G
Loft Surface	Shift+K	Shift+K
Extract	Shift+L	Shift+L
Create Contours	Shift+H	Shift+H
Project	Shift+; (semi-colon)	Shift+; (semi-colon)
Analysis	Shift+' (apostrophe)	Shift+A

Visualization Tool Set

Tool	Windows	Mac
Flyover	Shift+C	Shift+C
Walkthrough	Shift+U	Shift+U
Translate View	Shift+V	Shift+V
Rotate View	Shift+W	Shift+W
Light	Shift+Z	Shift+Z
Attribute Mapping	Shift+A	Option+A

Dims/Notes Tool Set

Tool	Windows	Mac
Constrained Linear Dimension	N	N
Unconstrained Linear Dimension	M	M
Angular Dimension	. (period)	. (period)
Radial Dimension	, (comma)	, (comma)

Walls Tool Set

Tool	Windows	Mac
Wall	9	9
Round Wall	Alt+9	Option+9
Wall Join	Alt+J	Option+J

Command Shortcuts (Modify in Workspace Editor)

File Menu

Command	Windows	Mac
New	Ctrl+N	Cmd+N
Open	Ctrl+O	Cmd+O
Close	Ctrl+W	Cmd+W
Save	Ctrl+S	Cmd+S
Page Setup	Ctrl+Alt+P	Cmd+Option+P
Print	Ctrl+P	Cmd+P
Exit	Alt+F4 or Ctrl+Q	Cmd+Q

Edit Menu

Command	Windows	Mac
Undo	Ctrl+Z	Cmd+Z
Redo	Ctrl+Y	Cmd+Y
Cut	Ctrl+X	Cmd+X
Copy	Ctrl+C	Cmd+C
Paste	Ctrl+V	Cmd+V
Paste in Place	Ctrl+Alt+V	Cmd+Option+V
Duplicate	Ctrl+D	Cmd+D
Duplicate Array	Ctrl+Shift+Alt+D	Cmd+Shift+Option+D
Select All	Ctrl+A	Cmd+A

View Menu

Command	Windows	Mac
Zoom–Normal Scale	Ctrl+3	Cmd+3
Zoom–Fit to Page Area	Ctrl+4	Cmd+4
Zoom–Fit to Objects	Ctrl+6	Cmd+6
Class Options–Active Only	Ctrl+Shift+Alt+3	Cmd+Shift+Option+3
Class Options–Gray Others	Ctrl+Shift+Alt+4	Cmd+Shift+Option+4
Class Options–Gray/Snap Others	Ctrl+Shift+Alt+5	Cmd+Shift+Option+5
Class Options–Show Others	Ctrl+Shift+Alt+6	Cmd+Shift+Option+6
Class Options–Show/Snap Others	Ctrl+Shift+Alt+7	Cmd+Shift+Option+7
Class Options–Show/Snap/Modify Others	Ctrl+Shift+Alt+8	Cmd+Shift+Option+8
Layer Options–Active Only	Ctrl+Alt+3	Cmd+Option+3
Layer Options–Gray Others	Ctrl+Alt+4	Cmd+Option+4
Layer Options–Gray/Snap Others	Ctrl+Alt+5	Cmd+Option+5

View Menu

Layer Options–Show Others	Ctrl+Alt+6	Cmd+Option+6
Layer Options–Show/Snap Others	Ctrl+Alt+7	Cmd+Option+7
Layer Options–Show/Snap/Modify Others	Ctrl+Alt+8	Cmd+Option+9
Standard views: Top/Plan	Ctrl+5	Cmd+5
Rendering–Wireframe	Ctrl+Shift+W	Cmd+Shift+W
Rendering–Sketch	Ctrl+Shift+S	Cmd+Shift+S
Rendering–Sketch Options	Ctrl+Shift+Alt+S	Cmd+Shift+Option+S
Rendering–OpenGL	Ctrl+Shift+G	Cmd+Shift+G
Rendering–Final Quality Renderworks	Ctrl+Shift+F	Cmd+Shift+F
Rendering–Hidden Line	Ctrl+Shift+E	Cmd+Shift+E
Rendering–Dashed Hidden Line	Ctrl+Shift+D	Cmd+Shift+D
Rendering–Final Shaded Polygon	Ctrl+Shift+P	Cmd+Shift+P
Set 3D View	Ctrl+0 (zero)	Cmd+0 (zero)
Unified View	Ctrl+Alt+L	Cmd+Option+L
Create Multiple Viewports	Ctrl+Shift+M	Cmd+Shift+M
Next View	Ctrl+Shift+. (period)	Cmd+Shift+. (period)
Previous View	Ctrl+Shift+, (comma)	Cmd+Shift+, (comma)

Modify Menu

Command	Windows	Mac
Move	Ctrl+M	Cmd+M
Move 3D	Ctrl+Alt+M	Cmd+Option+M
Send to Front	Ctrl+F	Cmd+F
Send Forward	Ctrl+Alt+F	Cmd+Option+F
Send to Back	Ctrl+B	Cmd+B
Send Backward	Ctrl+Alt+B	Cmd+Option+B
Align to Grid	Ctrl+- (hyphen)	Cmd+- (hyphen)
Align/Distribute	Ctrl+=	Cmd+=
Align/Distribute 3D	Ctrl+Shift+=	Cmd+Shift+=
Rotate Left 90	Ctrl+L	Cmd+L
Rotate Right 90	Ctrl+Shift+R	Cmd+Shift+R
Flip Horizontal	Ctrl+Shift+H	Cmd+Shift+H
Flip Vertical	Ctrl+Shift+V	Cmd+Shift+V
Set Working Plane	Ctrl+ \	Cmd+ \
Trim	Ctrl+T	Cmd+T
Join	Ctrl+J	Cmd+J

Modify Menu

Join and Fillet	Ctrl+Alt+J	Cmd+Option+J
Join (no Trim)	Ctrl+Shift+J	Cmd+Shift+J
Add Surface	Ctrl+Shift+Alt+A	Cmd+Shift+Option+A
Clip Surface	Ctrl+Shift+Alt+C	Cmd+Shift+Option+C
Intersect Surface	Ctrl+Shift+Alt+I	Cmd+Shift+Option+I
Compose	Ctrl+Shift+Alt+P	Cmd+Shift+Option+P
Convert to Group	Ctrl+K	Cmd+K
Convert to Mesh	Ctrl+Alt+R	Cmd+Option+R
Convert to 3D Polys	Ctrl+Alt+O	Cmd+Option+O
Convert to NURBS	Ctrl+Alt+N	Cmd+Option+N
Group	Ctrl+G	Cmd+G
Ungroup	Ctrl+U	Cmd+U
Edit <editing mode>	Ctrl+[	Cmd+[
Exit <editing mode>	Ctrl+]	Cmd+]
Show Guides	Ctrl+Alt+G	Cmd+Option+G
Hide Guides	Ctrl+Shift+Alt+G	Cmd+Shift+Option+G
Unrotate 3D Objects	Ctrl+Shift+0 (zero)	Cmd+Shift+0 (zero)

Model Menu

Command	Windows	Mac
Add Solids	Ctrl+Alt+A	Cmd+Option+A
Subtract Solids	Ctrl+Alt+S	Cmd+Option+S
Intersect Solids	Ctrl+Alt+I (uppercase i)	Cmd+Option+I (uppercase i)
Section Solids	Ctrl+Alt+T	Cmd+Option+T
Extrude	Ctrl+E	Cmd+E
Multiple Extrude	Ctrl+Alt+E	Cmd+Option+E
Extrude Along Path	Ctrl+Alt+X	Cmd+Option+X
Tapered Extrude	Ctrl+Alt+U	Cmd+Option+U
Sweep	Ctrl+Alt+W	Cmd+Option+W
Create Fillet Surface	Ctrl+Alt+Y	Cmd+Option+Y
Create Planar Caps	Ctrl+Alt+Q	Cmd+Option+Q
Create Surface from Curves	Ctrl+Alt+K	Cmd+Option+K

Spotlight Menu

Command	Windows	Mac
Number Instruments	Ctrl+Shift+Alt+N	Cmd+Shift+Option+N
Auto Number Positions	Ctrl+Shift+Alt+O	Cmd+Shift+Option+O
Focus Instruments	Ctrl+Shift+Alt+F	Cmd+Shift+Option+F
Replace Instrument	Ctrl+Shift+Alt+R	Cmd+Shift+Option+R
Find and Modify	Ctrl+Shift+Alt+M	Cmd+Shift+Option+M
Refresh Instruments	Ctrl+/	Cmd+/
Label Legend Manager	Ctrl+Shift+Alt+L	Cmd+Shift+Option+L

Tools Menu

Command	Windows	Mac
Organization	Ctrl+Shift+O	Cmd+Shift+O
User Origin	Ctrl+9	Cmd+9
SmartCursor Settings	Ctrl+8	Cmd+8
Plug-in Manager	Ctrl+Shift+Z	Cmd+Shift+Z
Workspace Editor	Ctrl+Shift+Alt+W	Cmd+Shift+Option+W

Text Menu

Command	Windows	Mac
Format Text	Ctrl+Shift+T	Cmd+Shift+T

Window Menu

Command	Windows	Mac
Palettes–Snapping	Ctrl+Shift+C	Cmd+Shift+C
Palettes–Attributes	Ctrl+Shift+A	Cmd+Shift+A
Palettes–Object Info	Ctrl+I (uppercase i)	Cmd+I (uppercase i)
Palettes–Working Planes	Ctrl+Shift+K	Cmd+Shift+K
Palettes–Resource Browser	Ctrl+R	Cmd+R
Palettes–Navigation	Ctrl+Shift+N	Cmd+Shift+N
Palettes–Visualization	Ctrl+Shift+Alt+V	Cmd+Shift+Option+V
Palettes– Activate Object Info Palette	Ctrl+‘ (backquote)	Cmd+Option+C

Other Shortcuts (Modify in Workspace Editor)

Cycle Through Modes on Tool Bar

1st group	U
2nd group	I (uppercase i)
3rd group	O
4th group	P
5th group	[
6th group	]

Miscellaneous Keys

Close 2D Polygon, Polyline, Polygonal Wall Network, or Path-based Objects	K
Toggle SmartCursor Cues	Y
Toggle Angles Relative to Prior Segment	/
Set Datum	G
Snap Loupe	Z
Set Smart Point, Edge, or Vector Snap Lock	T
Toggle Automatic Working Plane	\
Suspend Snapping	‘ (backquote)
X-ray Select mode	B
Coincident Selection	J

Toggle Snapping Palette Settings

Snap to Grid	A (AA = Set Grid dialog box)
Snap to Angle	S (SS = Angle Snaps dialog box)
Smart Points	D (DD = Smart Points dialog box)
Smart Edge (2D only)	F (FF = Edge Snaps dialog box)
Snap to Working Plane (3D only)	F
Snap to Object	Q (QQ = 3D Snapping dialog box)
Snap to Intersection	W
Snap to Distance	E (EE = Snap Distance dialog box)
Snap to Tangent (2D only)	R

Arrow Key Shortcuts (Modify in Vectorworks Preferences, Edit Tab)

Function	Windows	Mac
Switch active layer or class	Ctrl+Arrow (up and down for layers, left and right for classes)	Cmd+Arrow (up and down for layers, left and right for classes)
Pan drawing	Arrow	Arrow
Nudge objects	Shift+Arrow	Shift+Arrow
Move objects	Shift+Ctrl+Arrow	Shift+Cmd+Arrow

Worksheet Key Functions (Cannot be Modified)

Function	Windows	Mac
Move one cell up, down, left, or right	Arrow keys	Arrow keys
Move right by one cell	Tab	Tab
Move left by one cell	Shift+Tab	Shift+Tab
Move down by one cell	Enter	Enter
Move up by one cell	Shift+Enter	Shift+Enter

Numeric Key Pad (Cannot be Modified)

Switch to a Standard View

Top/Plan or Rotated Top/Plan	0 (zero)
Left Isometric	1
Front	2
Right Isometric	3
Left	4
Top	5
Right	6
Left Rear Isometric	7
Back	8
Right Rear Isometric	9

Miscellaneous Key Shortcuts (Cannot be Modified)

Function	Windows	Mac
Temporarily activate another tool	Press and hold Space bar, while you select and use another tool	Press and hold Space bar, while you select and use another tool
Temporarily activate Selection tool	Press and hold Alt while you select objects (Rotate, Mirror, Offset, Clip, and Move by Points tools)	Press and hold Cmd while you select objects (Rotate, Mirror, Offset, Clip, and Move by Points tools)
Temporarily activate Flyover tool	Press and hold Ctrl+mouse wheel button	Press and hold Ctrl+mouse wheel button
Temporarily activate Zoom tool	Press and hold Space bar+Ctrl (zoom in) or Space bar+Ctrl+Alt (zoom out)	Press and hold Space bar+Cmd (zoom in) or Space bar+Cmd+ Option (zoom out)
Hide unselected objects during movement	Press and hold Ctrl (Flyover, Walkthrough, Translate View, and Rotate View tools)	Press and hold Cmd (Flyover, Walkthrough, Translate View, and Rotate View tools)
Increase zoom by a factor of 2	Ctrl+1	Cmd+1
Decrease zoom by a factor of 2	Ctrl+2	Cmd+2
Increase zoom by a factor of 4	Ctrl+Alt+1	Cmd+Option+1
Decrease zoom by a factor of 4	Ctrl+Alt+2	Cmd+Option+2
Increase zoom by a factor of 2	CC	CC
Deselect all selected objects	XX	XX
Cancel the current operation	Esc	Esc
Exit text editing mode	Esc	Esc
Exit Group	Esc Esc	Esc Esc
While drawing walls, NURBS, etc., undo the last clicked point	Backspace	Delete
Display additional tooltip information	With tooltip displayed, press and hold Shift	With tooltip displayed, press and hold Cmd

Mouse Shortcuts (Cannot be Modified)

Function	Windows	Mac
Refresh the drawing view	Double-click Pan tool	Double-click Pan tool
Set the print page origin to the user origin	Double-click Move Page tool	Double-click Move Page tool
Increase zoom by a factor of 2	Double-click Zoom tool	Double-click Zoom tool
Decrease zoom by a factor of 2	Alt-double-click Zoom tool	Option-double-click Zoom tool
Open or edit a Resource Browser object	Ctrl-double-click the resource name or thumbnail	Cmd-double-click the resource name or thumbnail
Activate a class, design layer, or sheet layer	Double-click the item name in the Navigation palette	Double-click the item name in the Navigation palette
Switch to a saved view	Double-click the view name in the Navigation palette	Double-click the view name in the Navigation palette
Change visibility for all classes or layers	Alt-click a Visibility column in the Navigation palette	Option-click a Visibility column in the Navigation palette
Edit a saved view	Alt-double-click the view name in the Saved Views palette	Option-double-click the view name in the Saved Views palette
Activate Selection tool	Double-click anywhere in the drawing (works for many, but not all tools)	Double-click anywhere in the drawing (works for many, but not all tools)
Duplicate an object	With Selection tool selected, Ctrl-click and drag an object	With Selection tool selected, Option-click and drag an object
Create a similar object (activate tool and attributes)	Ctrl+Alt-click an existing object	Cmd+Option-click an existing object
Switch to Next or Previous view in history	On Windows, with a 5-button mouse, click button 4 or 5 (same as Back or Forward for web browsers)	*not applicable*
Pan the drawing view	With a wheel mouse, press and hold the wheel button	With a wheel mouse, press and hold the wheel button
Zoom in and out*	With a wheel mouse, roll forward to zoom in and backward to zoom out	With a wheel mouse, roll forward to zoom in and backward to zoom out
Scroll*	With a wheel mouse, Ctrl-roll (up and down); Shift-roll (left and right)	With a wheel mouse, Option-roll (up and down); Shift-roll (left and right)

**The mouse wheel zooms by default, and scrolls with a modifier key. You can swap these functions in the Vectorworks Preferences, Edit tab.*

Shortcuts Reserved by Operating System

Mac	Windows
Cmd+H	Shift+Alt+- (hyphen)
Cmd+Shift+Q	Alt+- (hyphen)
Cmd+Shift+1	Shift+Alt+P
Cmd+Shift+2	
Cmd+Shift+3	
Cmd+Shift+4	
Cmd+Shift+5	
Cmd+Shift+6	
Cmd+Shift+7	
Cmd+Shift+8	
Cmd+Shift+9	
Cmd+Option+8	
Cmd+Option+- (hyphen)	
Cmd+Option+=	
Cmd+Option+D	
Cmd+Option+H	
Cmd+Shift+Option+Q	