

Reference Guide

Product Release 32 December 2013



© CAST Group of Companies Inc., 2002-2013 All rights reserved. WYSIWYG, CAST Software, WYSIWYG Report, WYSIWYG Perform, WYSIWYG Design and Autofocus are trademarks of the CAST Group of Companies Incorporated. All other trademarks and logos are the property of their respective owners. Depending on your product/version, CAST incorporates source code or libraries which are licensed to CAST and copyright protected. For more information, go to http://www.cast-soft.com/content/third-party-libraries .



Summary of changes

Change location	Description of change
Chapter 7, page 84	Added Volumetric Beam mode
Chapter 8, page 138	Added Room Builder tool
Chapter 8, page 148	Added Rectangle tool
Chapter 8, page 147	Added Elliptical arc tool
Chapter 8, page 152	Added Linear Scale tool
Chapter 8, page 154	Added Wall tool
Chapter 8, page 156	Added Hatching styles for Walls (and Room Builder tool)
Chapter 8, page 199	Added Light Emission Enhancements / Per Element Static Light Emission
Chapter 8, page 216	Added Reflections in Shaded Views
Chapter 8, page 236	Added New and Enhanced Array tools
Chapter 8, page 240	Added Distribute between two points
Chapter 8, page 253	Added Revolve tool
Chapter 8, page 260	Added Curved Pipes
Chapter 10, page 389	Added Instruction tooltip

1 Introduction 1
- About this manual . . . 3
- Troubleshooting . . . 4
- Welcome to WYSIWYG . . . 6

2 The WYSIWYG suite of software products 7
- WYSIWYG Report . . . 9
- WYSIWYG Design . . . 10
- WYSIWYG Perform . . . 12
- WYSIWYG Perform - Console Edition . . . 14
- WYSIWYG Learn . . . 16
- WYSIWYG Network . . . 18

3 Installation, configuration, activation, and registration 19
- Installation . . . 21
- Customer Experience Improvement Program (CEIP) . . . 30
- Configuration, activation, and registration . . . 31
- Dongle Reauthorization . . . 35
- Upgrades . . . 38
- Membership and Leases . . . 39
- Passwords . . . 44

4 WYSIWYG user interface 47
- Navigation and screen set-up . . . 49
- WYSIWYG User Data Profile . . . 53
- Shortcut bars . . . 55
- Toolbars . . . 59
- Status bar . . . 68
- Wireframe views . . . 71
- Shaded views . . . 78

5 CAD mode 95
- Layout tabs . . . 97
- The CAD environment . . . 98
- The Library Browser . . . 133
- Drawing objects . . . 138
- Editing objects . . . 185
- CAD tools . . . 226
- Hang structures . . . 257
- Hanging and focusing fixtures . . . 274
- Color, gobos, and accessories . . . 315
- LED Wizard . . . 322
- Video Manager . . . 335
- Image Manager . . . 343
- The Flight Case . . . 351
- Rendering . . . 354

6 Data mode 355
- Layout tabs . . . 357
- Spreadsheets . . . 358
- Patch . . . 366
- Errors . . . 376

7 Managing your WYSIWYG document 379
- File menu . . . 381

Options menu . . . 389
WYSIWYG Viewer . . . 410

8 Presentation mode 411
Layout tabs . . . 413
Reports . . . 414
New Plots . . . 418
Layouts . . . 436
Images . . . 463
Worksheets . . . 464
Pipe tapes . . . 467
Production Team Info Wizard . . . 469

9 Design mode 471
Layout tabs . . . 473
Design tools . . . 474
Fixture groups . . . 484
Palettes . . . 486
Looks . . . 487
Cuelists . . . 490
Camera Paths . . . 493
Rendering . . . 497
Background Rendering Manager . . . 508

10 Smart Cells 517
Smart Cells . . . 519

11 Live mode 523
Layout tabs . . . 525
Connecting to consoles . . . 526
Working with consoles . . . 530
Image Console . . . 535
AutoPatch . . . 539
Visualization . . . 541
Autofocus . . . 544
Rendering . . . 547
DMX Camera . . . 548
Patching camera paths . . . 550

A Using the SuperPro Monitoring Tool 553
SuperPro Monitoring Tool . . . 555

B Keyboard shortcuts and hot keys 559
Keyboard shortcuts and hot keys . . . 561

C Using AutoFocus with the HogPC 569
Setting up AutoFocus to work with the HogPC . . . 571

D Connecting to an ArtNet Device 575
Connecting to an ArtNet Device . . . 577

E Connecting to Luminair 579
Connecting to Luminair . . . 581

F **Using the 3D Connexion Mouse with WYSIWYG** **583**
Using the 3D Connexion Mouse 585

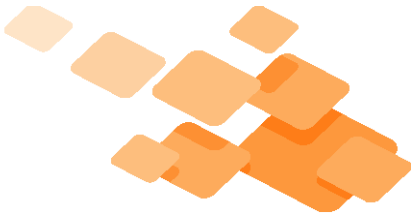
G **Using the WYSIWYG Dongle Update Utility** **587**
WYSIWYG Dongle Update Utility 589

H **Using the WYSIWYG Dongle Replacement Utility** **595**
WYSIWYG Dongle Replacement Utility 597

Index **602**

Chapter 1

Introduction



This chapter introduces you to the WYSIWYG Reference Guide, provides information for troubleshooting, and lists contact information if you need further assistance.

In this chapter

About this manual 3

Troubleshooting 4

Welcome to WYSIWYG 6



About this manual

Introduction

This manual provides instructions for the installation and operation of the different levels of the WYSIWYG application. WYSIWYG is available at the following levels:

- WYSIWYG Report
- WYSIWYG Design
- WYSIWYG Perform
- WYSIWYG Perform - Console Edition
- WYSIWYG Learn
- WYSIWYG Network

All levels are discussed in this manual. For a detailed description of the levels, refer to Chapter 2, “The WYSIWYG suite of software products.”

All the levels use the same user interface, described in Chapter 4, “WYSIWYG user interface.” However, you can perform different tasks in each mode. For example, you draw in CAD mode and edit in Data mode. These modes may have different features based on the level of WYSIWYG that you are running.

Text conventions

The following text conventions are used in this manual:

- Menus and menu commands appear in **Arial bold**. For example, “from the **Edit** menu, choose **Layers**.”
- User interface elements such as buttons, tools, shortcuts, and dialog boxes appear in *Tahoma Oblique*. For example, “to draw a riser, click the *Riser* tool on the *Draw* toolbar.”
- Keyboard keys are indicated in ALL CAPS. For example, “press TAB to enter the missing coordinate.”
- References to manuals appear in *italic* font. For example, “for additional information on working with recorded loads, please refer to your *Sensor CEM User Manual*.”

Troubleshooting

Introduction

If you have problems using your software, please consult the following resources:

- the contents of this manual or the online Help available from within WYSIWYG
- the WYSIWYG Web site at http://www.cast-soft.com/wysiwyg

If these resources are insufficient, please contact Technical Support.

Help from Technical Support

For assistance with a problem, contact Technical Support directly at one of the offices identified below.

When calling for help, please have the following information ready:

- the level of WYSIWYG that you are using (Report, Design, or Perform)
- software version (from the **Help** menu, choose **About WYSIWYG**)
- the kind of computer that you are using (processor and speed, amount of memory, graphics card, operating system)
- DMX interface type (if applicable)

North America

CAST Software

Technical Support

35 Ripley Avenue, Unit 1

Toronto, ON M6S 3P2

Canada

Phone: +1 877 989 2278

Fax: +1 416 597 9594

E-mail: techsupport@cast-soft.com

Web site: www.cast-soft.com

United Kingdom

A.C. Lighting Ltd.

Technical Sales & Support:

Centauri House

Hillbottom Road

High Wycombe

Bucks, HP12 4HQ

UK

Phone: +44 (0)1494 446000

Fax: +44 (0)1494 461024

E-mail: wysiwyg@aclighting.com

Web site: www.aclighting.com

Germany

ARCUS GmbH

Obermeiers Feld 2-4

33104 Paderborn

Germany

Phone: +49 (0) 5254 93069 0

Fax: +49 (0) 5254 93069 29

E-mail: support@arcus-lighting.com

Web site: www.arcus-lighting.com

Japan

Theater Engineering

Phone: +81 (0) 3 3706 5611

Fax: +81 (0) 3 3706 6766

E-mail: info@thea-eng.co.jp

Web site: www.thea-eng.co.jp

Support for WYSIWYG Perform - Console Edition

For assistance with a problem related to WYSIWYG Perform - Console Edition, contact the manufacturer of the console to which you are connected. The console manufacturer is listed on the Welcome page that appears when you launch the application.



Welcome to WYSIWYG

Introduction

The WYSIWYG suite of software products, designed specifically for lighting professionals, offers a range of solutions to meet the needs of designers, assistants, electricians, console operators, teachers, and students.

WYSIWYG is the essential tool if you are looking to

- increase creative freedom
- save time
- win more contracts
- cut costs
- be more environmentally friendly

Within the WYSIWYG suite of products, any higher level product contains all the features of a lower level product. WYSIWYG Report, Design, and Perform are the core products, with WYSIYWG Report offering basic CAD and paperwork solutions, and WYSIWYG Perform providing an end-to-end solution including CAD, paperwork, rendering, and pre-visualization.

In less than one day of training, you can begin working with WYSIWYG and see your lighting design without actually installing any lights. You can even see what a new lighting fixture can do before you rent, buy, or install it. Once your design is complete, WYSIWYG handles the logistics; it automatically generates plots, schedules, pipe-tapes, and color and gobo lists.

For more information on each of the three main levels of WYSIWYG, see the appropriate section:

- **WYSIWYG Report** - For details, see “WYSIWYG Report” on page 9.
- **WYSIWYG Design** - For details, see “WYSIWYG Design” on page 10.
- **WYSIWYG Perform** - For details, see “WYSIWYG Perform” on page 12.

Chapter 2

The WYSIWYG suite of software products



This chapter describes the different levels of WYSIWYG.

In this chapter

WYSIWYG Report 9
WYSIWYG Design 10
WYSIWYG Perform 12
WYSIWYG Perform - Console Edition 14
WYSIWYG Learn 16
WYSIWYG Network 18

WYSIWYG Report

Introduction

WYSIWYG Report is the basic WYSIWYG package. It allows lighting professionals to create 2D plans of their productions along with all the necessary paperwork.

WYSIWYG Report is also available in an educational edition. This product, WYSIWYG Report Educational Edition, is a fully functional version of Report that is available to students and faculty at a special price.

Note: The following specifications are subject to change.

Software features

- 2D plotting of set and lighting designs (plans, sections and elevations)
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, and so on
- Extensive CAD tools
- Excellent zooming and maneuvering abilities
- Custom layers and scenes for complete control and management of information
- Loading and exporting of DXF, DWG, SketchUp, Excel, BMP, and JPG file types
- Tabular editing of lighting information including channel, dimmer, color, and so on
- Fully integrated paperwork and 2D CAD systems
- Error checking of production paperwork and hookups
- Reports including instrument counts and schedules, patch sheets, and so on
- Printing of plots and reports up to 36”x 48” / A0
- Custom report and plot content and layout
- Ability to print all view types including 2D CAD, reports, spreadsheets, bitmap files, jpeg files, and logos
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.

Modes

The following working modes are available in WYSIWYG Report:

- CAD 2D
- DATA
- PRES 2D

Refer to the respective chapters of this guide for detailed explanations about the features in each mode.

WYSIWYG Design

Introduction

WYSIWYG Design builds on the feature set of Report and adds the ability for lighting professionals to explore their designs in a 3D environment. 3D CAD tools, shaded and isometric views, and photo-realistic rendering tools are added, as well as the ability to create and store images of lighting looks using the integrated design tools.

WYSIWYG Design is also available in an educational edition. This product, WYSIWYG Design Educational Edition, is a fully functional version of Design that is available to students and faculty at a special price.

Note: The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report
- 3D plotting of set and lighting designs
- Multiple working views including isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, and so on
- Extended library includes textures and materials
- Library objects are multi-layered for realistic texturing
- Objects have variable translucency
- Fully integrated paperwork and CAD systems
- Loading and exporting of DXF, DWG, SketchUp, Excel, BMP, and JPG file types
- Print all view types including 3D views, bitmap files, jpeg files, spreadsheets and logos
- Integrated design mode allows creation of static lighting looks without a console
- Rendering generates a photo-realistic picture of your lighting look
- Rendering wizard provides options for smoke, shadows, ambient light, and so on
- Renderings include bounce and reflection from surfaces
- Background Rendering Manager that enables you to queue up multiple renderings to run in the background without even requiring WYSIWYG to run
- OpenGL technology for improved beam simulation
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The following working modes are available in WYSIWYG Design:

- CAD
- DATA
- DESIGN
- PRES

Refer to the respective chapters of this guide for detailed explanations about the features in each mode.

WYSIWYG Perform

Introduction

WYSIWYG Perform builds on the feature set of Report and Design and adds the ability for lighting professionals to pre-cue entire shows in real time. Users can create their productions in 3D along with all the necessary paperwork. A console can then be connected to the computer and WYSIWYG Perform will simulate the exact effect of the console's output in real-time.

Note: The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report and Design
- 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom, and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, and so on
- Fully integrated paperwork and CAD systems
- Live mode for real-time lighting visualization
- DMX camera to allow you to dynamically reposition your point of view in Live mode.
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- LED grids allow simulation of lasers, neon, LEDs, projection screens, and so on
- Image Console enables you to map an image to patched grids of light-emitting surfaces and control its intensity in Live mode
- Easy hook-up to consoles and other DMX sources
- Auto Focus protocol lets WYSIWYG send color and focus data to consoles
- Integrated design mode allows you to create lighting looks without a console
- Rendering generates a photo-realistic picture of your cue or lighting look
- Renderings include bounce and reflection from surfaces
- Background Rendering Manager that enables you to queue up multiple renderings to run in the background without even requiring WYSIWYG to run
- Animated renderings available though certified service providers
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The following working modes are available in WYSIWYG Perform:

- CAD
- DATA
- DESIGN
- PRES
- LIVE

Refer to the respective chapters of this guide for detailed explanations about the features in each mode.



WYSIWYG Perform - Console Edition

Introduction

WYSIWYG Perform - Console Edition is a version of WYSIWYG Perform intended for distribution with a specific console or family of consoles. Users purchasing a compatible console will receive this edition of WYSIWYG Perform to create shows and begin pre-cueing activities. The Console Edition of WYSIWYG Perform includes most of the functionality of Perform with certain features restricted or removed.

Note: The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Perform, other than printing or exporting in any format
- 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom, and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, and so on
- Fully integrated paperwork and CAD systems
- Live mode is restricted to visualizing one patch universe at a time
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- LED grids allow simulation of lasers, neon, LEDs, projection screens, and so on
- AutoPatch transmits the entire show patch regardless of channel count
- Rendering is available in Design and Live modes but is limited to 76,800 pixels

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Note: You cannot upgrade from WYSIWYG Report or Design to WYSIWYG Perform - Console Edition. You can, however, upgrade from WYSIWYG Perform - Console Edition to WYSIWYG Perform, removing any console restrictions.

Modes

The following working modes are available in WYSIWYG Perform - Console Edition:

- CAD
- DATA
- DESIGN
- PRES
- LIVE

Refer to the respective chapters of this guide for detailed explanations about the features in each mode.

WYSIWYG Learn

Introduction

WYSIWYG Learn is a special configuration of WYSIWYG that allows an educational institution to run ten instances of the software on a Local Area Network. The workstations can be configured to run any combination of WYSIWYG Report or WYSIWYG Design. In addition, one workstation can run WYSIWYG Perform.

Note: The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report, Design, or Perform
- 2D and 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom, and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, and so on
- Fully integrated paperwork and CAD systems
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- LED grids allow simulation of lasers, neon, LEDs, projection screens, and so on
- Easy hook-up to consoles and other DMX sources
- Auto Focus protocol lets WYSIWYG send color and focus data to consoles
- Rendering generates a photo-realistic picture of your cue or lighting look
- Renderings include bounce and reflection from surfaces
- Background Rendering Manager that enables you to queue up multiple renderings to run in the background without even requiring WYSIWYG to run
- Animated renderings are available though certified service providers
- Multi-monitor support

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The following working modes are available in WYSIWYG Learn:

- CAD
- DATA
- DESIGN
- PRES
- LIVE

Refer to the respective chapters of this guide for detailed explanations about the features in each mode.



WYSIWYG Network

Introduction

WYSIWYG Network allows you to run as many licenses of WYSIWYG Report, Design, and Perform as you want to over a network. The workstations can be configured to run any combination of these products with the use of a single network dongle.

Note: The following specifications are subject to change.

Software features

- Includes all the features of WYSIWYG Report, Design, and/or Perform
- 2D and 3D plotting of set and lighting designs
- Multiple working views including front, back, sides, top, bottom, and isometric
- Comprehensive 3D library of fixtures, accessories, truss, colors, gobos, props, and so on
- Fully integrated paperwork and CAD systems
- Accurate simulation of conventional and automated fixtures
- OpenGL technology for improved beam simulation
- LED grids allow simulation of lasers, neon, LEDs, projection screens, and so on
- Easy hook-up to consoles and other DMX sources
- Auto Focus protocol lets WYSIWYG send color and focus data to console
- Rendering generates a photo-realistic picture of your cue or lighting look
- Renderings include bounce and reflection from surfaces
- Background Rendering Manager that enables you to queue up multiple renderings to run in the background without even requiring WYSIWYG to run
- Animated renderings available though certified service providers
- Multi-monitor support
- 12 months of software and library updates included

Upgrade suggestions

- A faster processor improves responsiveness and rendering times.
- Memory requirements depend on the size of the show. Insufficient memory can seriously affect performance. Add memory if there is excessive disk activity.
- A high-performance OpenGL accelerator card significantly improves the performance of real-time simulation.

Modes

The working modes available in WYSIWYG Network depend on your product configuration. Refer to the WYSIWYG Report, Design, or Perform sections accordingly. Detailed explanations of each of the various modes are included in this guide.

Chapter 3

Installation, configuration, activation, and registration



This chapter provides instructions on how to install, configure, activate, and register the WYSIWYG software. You must activate and register the software before you can start using WYSIWYG.

In this chapter

Installation	21
Configuration, activation, and registration	31
Dongle Reauthorization	35
Upgrades	38
Membership and Leases	39
Passwords	44



Installation

Introduction

Follow the procedures in this section to install the level of WYSIWYG that you have purchased. The requirements are separated according to the level of functionality that you want to achieve (**Minimum** or **High Performance**).

Minimum requirements

The following requirements are separated according to the level of functionality that you want to achieve (Minimum, High Performance).

Attention:

- WYSIWYG must be installed on an NTFS file system.
- For Enhanced Simulation, a video card supporting OpenGL 2.1 is required; otherwise the software will disable Enhanced Simulation.
- When running WYSIWYG on a MAC with Bootcamp, performance is comparable to running WYSIWYG on a PC. When running WYSIWYG on a MAC in Parallels, performance is slightly degraded when compared to a PC.

Visualization Requirements

There are multiple levels of increasing beam simulation offered in WYSIWYG's Shaded Views.

- **Minimum Visualization Requirements**: The minimum video card requirements to display the default Enhanced Beams must support OpenGL 2.1.
- **Intermediate Visualization Requirements:** For higher level graphics offered by the Enhanced Visualization setting, your video card must support OpenGL 3.0.
- **High Performance Visualization Requirements**: To obtain the highest graphics level offered by Deferred Rendering with Enhanced Visualization, your video card must support OpenGL 3.3.

Note: Deferred Rendering is required to run Volumetric Beams. If this option is not available, it means the graphics card on your computer does not support Volumetric beams.

beams.**1. Minimum System Requirements (basic functionality)**

- Intel Core 2 Duo or compatible processor, 2.0 Ghz or better
- Windows XP with Service Pack 3, Windows Vista with Service Pack 2, Windows 7 with Service Pack 1 or Windows 8
- MAC Intel Processor platforms with Windows XP (SP3) or Windows Vista (SP2), Windows 7 (SP1) or Windows 8 running via Bootcamp (recommended) or Parallels
- 4GB RAM
- 1GB free disk space (NTFS file system)
- 1024 x 768 resolution
- OpenGL-accelerated “gaming-level” video card, with 256MB (or better) video memory and OpenGL 2.0 (minimum) support
- Free USB port

2. High Performance System Requirements (for enhanced visualization and rendering time)

- Quad-Core Intel Core i5/i7 or compatible processor, 2.4 Ghz or better
- Windows XP with Service Pack 3, Windows Vista with Service Pack 2, Windows 7 with Service Pack 1 or Windows 8; the 64bit version of Windows Vista/7/8 is not required, but strongly recommended
- MAC Intel Processor platforms with Windows XP (SP3) or Windows Vista (SP2), Windows 7 (SP1) or Windows 8, running via Bootcamp; Parallels not recommended
- 8GB RAM
- 1GB free disk space (NTFS file system)
- OpenGL-accelerated "gaming-level" video card, with 1GB (or better) video memory and OpenGL 2.0 (minimum) support; OpenGL 3.0 support required for Deferred Rendering simulation in Shaded Views (strongly recommended for performance gains)
- Free USB port

General requirements

In addition to the requirements listed above, note the following product-specific requirements:

WYSIWYG Perform

- WYG-it 2 DMX reception device (except when used with certain consoles - contact the console manufacturer for details)
- MIDI output port to use with AutoFocus

WYSIWYG Network/WYSIWYG Learn

- Network interface card running TCP/IP

For WYSIWYG Perform workstations:

- Use a WYG-it 2 or compatible DMX reception device (except when used with certain consoles - contact the console manufacturer for details)
- MIDI output port to use with Autofocus-enabled consoles

Installing WYSIWYG

The installation procedure for WYSIWYG is the same regardless of the level that you have purchased.

The WYSIWYG install panel appears when the CD is inserted into your PC's CD drive or when you double-click the install file (if you have downloaded the software). This guide displays buttons for all the installation options:

- Install WYSIWYG
- Install WYSIWYG Network Server (see "Installing WYSIWYG Learn or WYSIWYG Network" on page 24)
- Quick Start Guide
- Quick Start Tutorials
- Reference Guide
- Release Notes
- Install Adobe Acrobat Reader

To install WYSIWYG

1 Insert the installation CD into your PC's CD drive *OR* double-click the install file (if you have downloaded the software).

Result: The WYSIWYG Install panel appears.



Click *INSTALL WYSIWYG* to install the software. Follow the steps in the Installation Wizard as prompted.

Note: If you are installing WYSIWYG on a computer that will act as a combined Learn/Network Server and Client, after completing the WYSIWYG installation, simply attach the network dongle to it to activate the Server functionality.

Note: If you have an installation CD and the installation Wizard does not automatically open when you insert the CD into your computer:

a. Access the CD-ROM drive of your computer.

b. Double-click the icon for the *Wyginstall* program.

c. Follow the steps in the Installation Wizard.

Installing an In-Place update

If you already have a previous release of WYSIWYG installed on your computer, you can easily update to the current release by installing an in-place update. This update allows you to enhance a version of WYSIWYG that is a maximum of two releases back from the current release. You do not have to uninstall the previous version of WYSIWYG to perform the update.

To install an in-place update

Note: Before proceeding with updating your current WYSIWYG release with the in-place-update, ensure that the WYSIWYG software is not running on your computer.

1 From the *WYSIWYG Install* window, select *Install WYSIWYG*.

2 Select *Update the product selected below*.

3 From the list of WYSIWYG releases, highlight the release that you want to update, and then click *Next*.

4 On the *WYSIWYG Update* window, select *Next*.

5 On the *WYSIWYG Update* window, click *Yes* to accept the terms of the license.

Result: The updated files are copied to your computer.

6 After all the files are copied, click *FINISH* to complete the update.

Installing WYSIWYG Learn or WYSIWYG Network

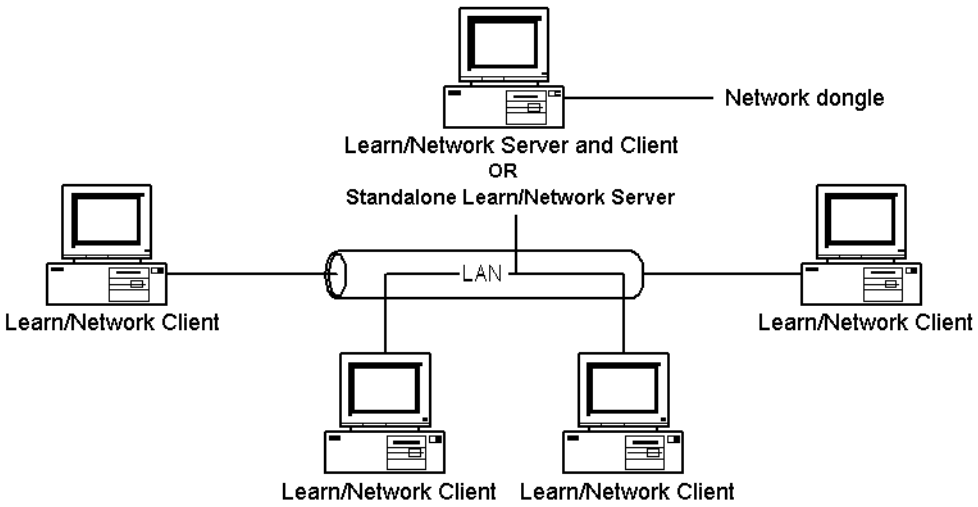
Software components

The WYSIWYG Network and WYSIWYG Learn systems comprise a computer that acts as the server and one or more clients. In your network, you need to have only one computer that acts as the server (the computer with the network dongle attached to it).

The server can be standalone, in which case WYSIWYG is not installed on it, or it can act as both server and client. In the former scenario, you install the WYSIWYG Network Server which installs the Sentinel Protection service. In this latter scenario, you install the full WYSIWYG program, which also includes the network server portion of the Sentinel Protection service. (As a result, in the latter scenario, installing the WYSIWYG Network Server should not be performed.) This Sentinel Protection service is required and its status should be started to detect the WYSIWYG Learn or Network dongle (a list of current services on a workstation can be viewed under *Control Panel > Administrative Tools > Services*).

The Learn or Network Client computers should have WYSIWYG installed on them, which also includes the network Sentinel Protection service. (As a result, the WYSIWYG Network Client is no longer required and should not be performed.) As in the case of the Server computer, the Sentinel Protection service enables the software to function over the network WYSIWYG Learn can run multiple clients (usually in groups of 10, 20, 30, 40 or 50) on a network. The number of clients for WYSIWYG Learn/Network is restricted based on the configuration purchased.

A typical WYSIWYG Network/Learn setup is illustrated on the following page. All computers running Network/Learn components must be on the same network (and therefore have the same network ID and subnet mask). It is also necessary that Server and Client computers have their firewall turned off. Following the installation, please ensure that the computers are rebooted before running the WYSIWYG Learn/Network configurations.



Network services

Windows XP

Services are supported in Windows operating systems. Services start running as soon as your computer is turned on, regardless of whether the user is logged in. The service begins automatically each time the workstation is restarted, and runs in the background without any prompting from the user. The service may also be started manually; the choice is available during the installation process.

Network protocol

WYSIWYG Learn or WYSIWYG Network will run on any LAN that supports TCP/IP protocol (NetBEUI and IPX/SPX networks are not currently supported).

Detecting the network dongle

If the service is not running, the Clients will not be able to detect the dongle and the application will terminate. (To check the Sentinel Protection service, check under the *Control Panel > Administrative Tools > Services*). Ensure that the dongle is attached to the workstation that is acting as the Server. Ensure that the firewall is turned off on all the computers running as the server computer and all client computers.

Installation options

Based on whether your Server computer is a combined Learn/Network Server *and* Client or a standalone Server, the installation varies slightly.

- If you intend to run WYSIWYG on the server computer (i.e., you want a *combined* Learn/Network Server/Client), then you must install the full WYSIWYG program on this computer. While this installation includes all required server files, they are only activated after you reboot the computer and then attach the network dongle to the Server. For details, see “To install WYSIWYG” on page 23.
- If you do not want to run WYSIWYG on the Server computer (i.e., you want to have a *standalone* Learn/Network Server), the installation is condensed. See “To install the Learn/Network Server (standalone)” below.

To install the Learn/Network Server (standalone)

Perform this installation if you want to configure a standalone Learn/Network Server (i.e., you do not want to run WYSIWYG on this computer). After the installation, simply reboot the computer and then attach the network dongle to it to activate the Server functionality.

Before you begin

- Previous versions of WYSILAB or WYSIWYG Learn/Network must be removed before beginning the install process (if this step is not completed, the install menu will offer the option before continuing with the installation). The workstations must be restarted for the changes to take effect.

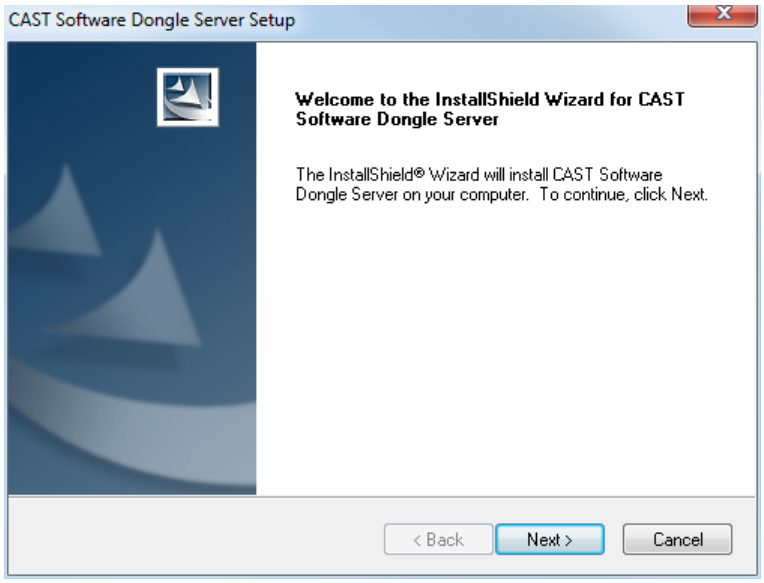
1 Insert the installation CD into your PC's CD drive *OR* double-click the install file (if you have downloaded the software).

Result: The WYSIWYG Install panel appears.

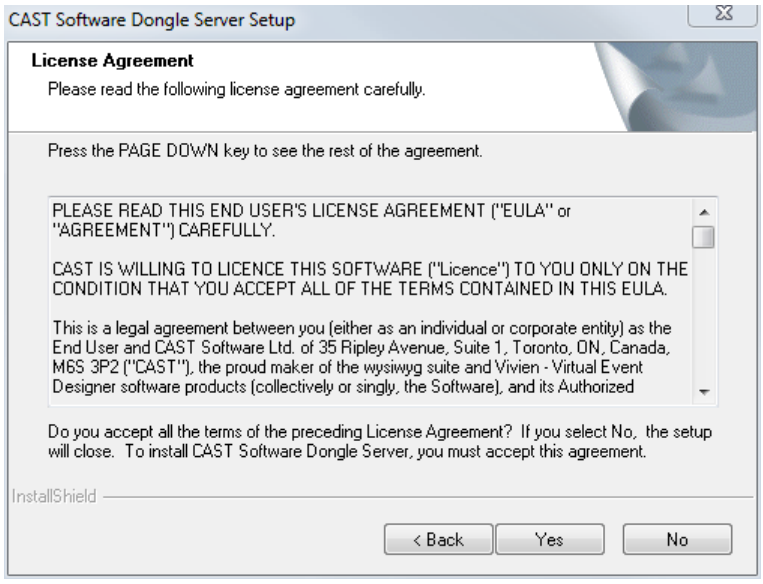


2 Select *Install WYSIWYG Network Server*.

Note: The Server should only be installed on one computer on the network. This is the computer to which you will attach the network dongle.

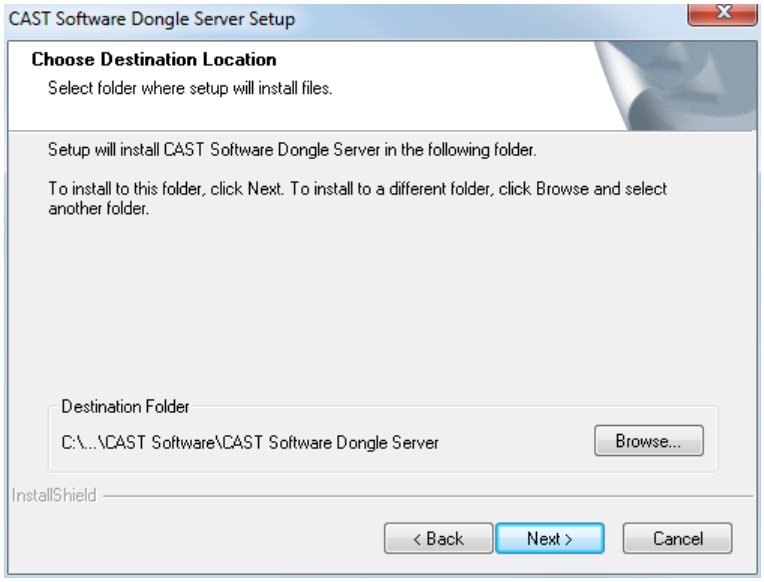


3 On the *Setup* window, select *Next* to continue the install process.



4 Click *Yes* to accept the terms of the license.

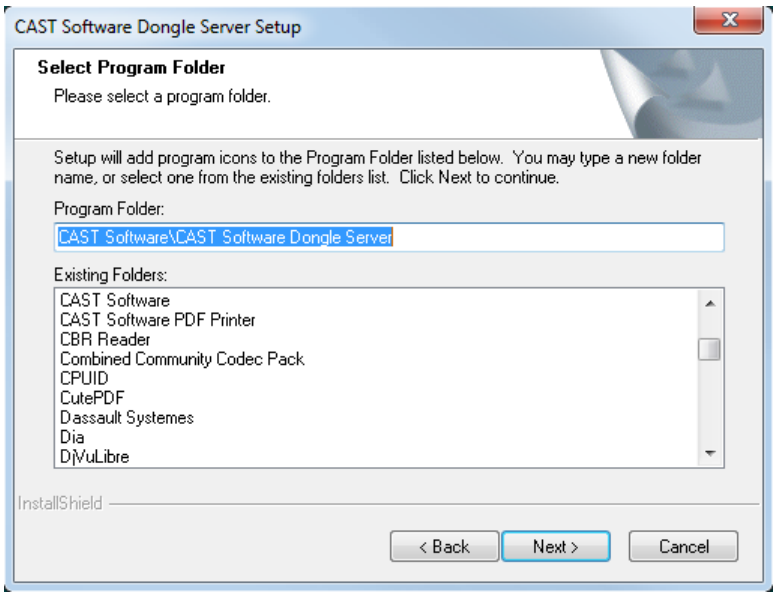
Result: The *Choose Destination Location* window appears.



5 Select a destination location for the Server files. The default location is C:\Program Files\WYSIWYG Server.

6 Click *Next.*

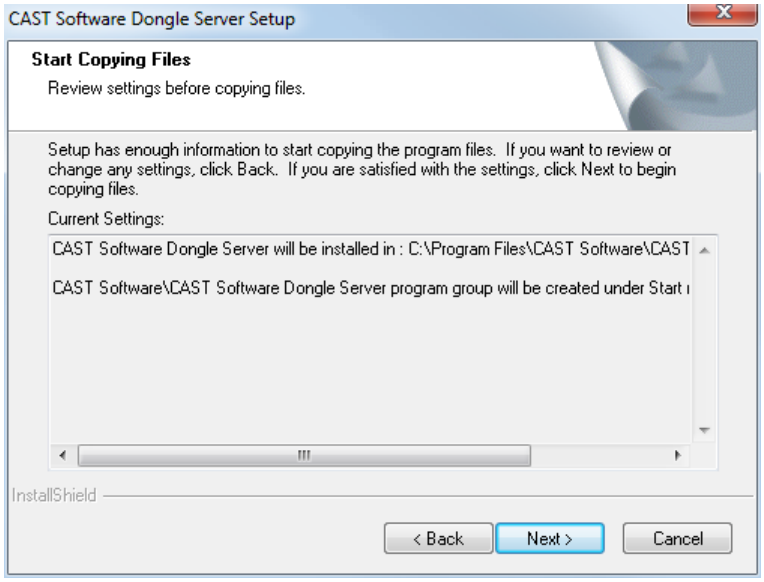
Result: The *Select Program Folder* window appears.



7 Select a program folder for the shortcut. The default is WYSIWYG Server.

8 Click *Next*.

Result: The *Start Copying Files* window appears.



9 Review the installation settings, and then click *Next* to begin the installation.

10 After all the files are copied, click *FINISH* to complete the installation.

To install the Network/Learn Client

1 Insert the installation CD into your PC's CD drive *OR* double-click the install file (if you have downloaded the software).

Result: The WYSIWYG Install panel appears.



2 Click *Install WYSIWYG.*

Result: The CAST license screen appears.

3 If you accept the terms of the license, click *Yes* to continue. The installation begins.

4 Click *FINISH* to complete the installation.



Customer Experience Improvement Program (CEIP)

In order to better serve our customers, and to enable our products to meet our customers ongoing and growing needs, we regularly engage in various methods to gather client feedback, including direct responses to support and problem requests, as well as various other comments and queries. We encourage our customers to participate in giving feedback in order to get the most out of our products and create positive experiences. However, given the large scope of our customer base, it is impossible to reach out to all our customers directly.

Our Customer Experience Improvement Program (CEIP) is a new way to allow all our customers to contribute to the features, design and development of CAST Software products. This program enables our customers to provide us with various information, including information about the hardware configuration, the features you use most (and least), and the nature of the problems you face. Based on this information, we will be able to improve our products and the features that you use the most.

We will not collect any personal data, such as your name, address, phone number, or keyboard input.

We are convinced that the CEIP will provide valuable anonymous information that will lead to software improvements and enhanced functionality to better meet the needs of our customers.

We use the service EQATEC Analytics to provide the technology needed to implement our CEIP program. You can read more at http://www.cast-soft.com/content/ceip



Configuration, activation, and registration

Introduction

After installing the product, when you launch WYSIWYG for the first time, the Product Configuration Wizard appears, enabling you to choose the product level that you want to run. You can choose from standard product levels (such as Design and Report), Network products, or you can run the Demo or Viewer versions of WYSIWYG.

Note: To run any of the products listed in the Wizard, you must first have the appropriate dongle installed on your computer (to run a network product, you must have a network dongle installed on a computer that is accessible from your PC).

After you proceed through the steps of the Wizard, you are prompted to activate your product (only if you are upgrading from version 3.X to the latest version), and then you are prompted to register the software. You must activate and register the WYSIWYG software before you can start using it. To activate the WYSIWYG software involves entering an 11-digit activation code. To register the software involves entering a validation code.

This section is broken down into the following three steps:

- starting WYSIWYG for the first time and running through the Product Configuration Wizard
- activating the software
- registering the software

To start WYSIWYG for the first time

Note: Before you perform this procedure, ensure that you have the correct dongle installed on your computer, or that the network dongle is installed on a computer that you can access from your PC.

1 On your desktop, click the shortcut icon to start WYSIWYG (or click **Start** > **All Programs** > **CAST Software** > **WYSIWYG Release ##**).

Result: The WYSIWYG Product Configuration Wizard appears.

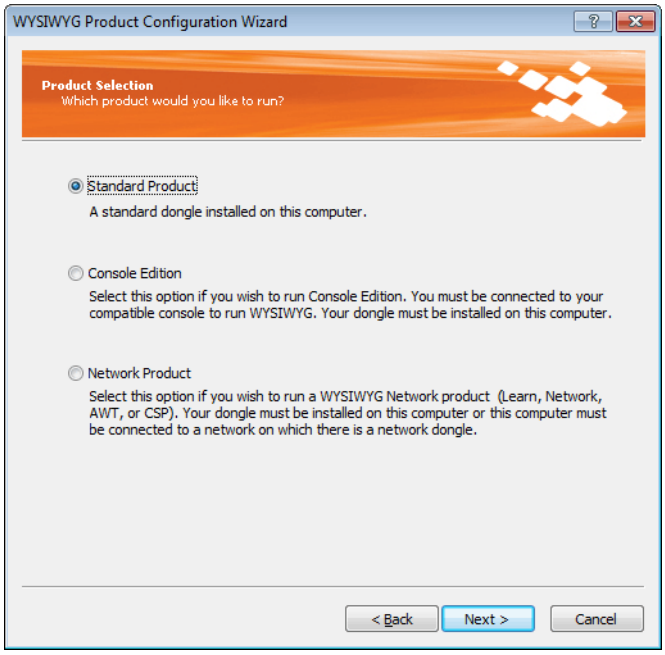


2 To ensure that the Wizard appears the next time you launch WYSIWYG, leave the check mark in the box beside *Show me this wizard next time I run WYSIWYG*; to configure the program so the Wizard does not appear the next time you launch WYSIWYG, clear this check box.

Note: If you clear this check box, you can always set it again in the Application Options window. For details, see page 390.

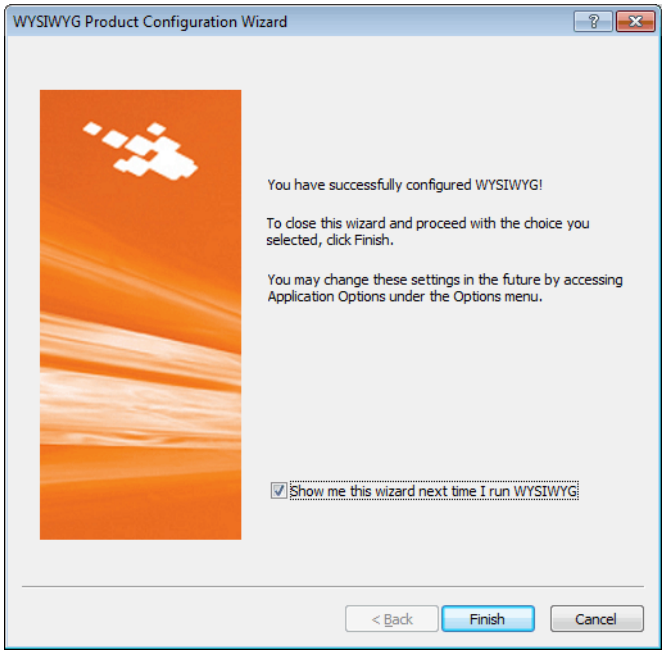
3 Click *Next*.

Result: The Product Selection window appears, enabling you to choose the product level that you want to launch.



4 Select the product that you want to launch (you must have the correct dongle installed to launch the program), and then click *Next*.

Result: The program verifies that you have the correct dongle attached, and the Product Configuration Successful window appears:



5 Click *Finish*.

6 The product that you chose launches and the Activate or Register window appears (based on the level of product that you have purchased).

- If you have purchased a new box product containing a new version dongle skip to the product registration process. For details, see “To register WYSIWYG” on page 33.
- If you are upgrading from WYSIWYG 3.X, please contact CAST Technical Support.

To register WYSIWYG

1 Once you have gone through the Product Configuration Wizard and you have successfully activated your product, the Register window appears, prompting you to register your software.



Note: You can postpone registration for a period of up to 15 days after installation.

To register later, click *Register Later*. You must register and obtain a Registration Validation Code to continue using the product after this time. It is recommended that you register as soon as possible to avoid any restrictions on using the product.

You can register by phone, online, fax, or e-mail.

2 To register by phone, click *Phone*, and then proceed to step 3.

To register by any of the other methods, select the appropriate button and follow the instructions as prompted.

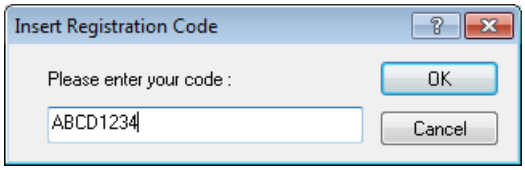
When you have obtained your validation code, click *I have my validation code*, and then proceed to step 4.

3 If you have chosen to register by phone, the *Register by Phone* dialog box is displayed.



Follow the instructions in the dialog. Type your validation code, and then click *Register*. A successful registration completes the registration and activation process.

4 If you selected *I have my validation code* in step 2, the *Insert Registration Code* dialog box appears.



5 Type your validation code, and then click *OK*. A successful registration completes the registration and activation process.



Dongle Reauthorization

Introduction

Reauthorization requires that all WYSIWYG Members "reauthorize" their dongle periodically by logging into the WYSIWYG Members site (http://www.cast-soft.com/wysiwyg/), getting their Dongle Reauthorization Code, and then applying this code to the dongle.

This initiative will help safeguard your dongle, ensuring that if it is lost, stolen, or found by someone else, it will be useless beyond the reauthorization date.

Reauthorization codes and WYSIWYG releases

Starting with Release 24, all dongles will have a Reauthorization Expiry Date burned onto them as follows:

- Dongles that were registered with a version of WYSIWYG prior to Release 24 will have their Reauthorization Expiry Date set to 30 days forward from the time when Release 24 or later is first launched with such a dongle. Within these 30 days, you must follow the Reauthorization process described below or the dongle will cease to function. (It will resume functioning once you reauthorize the dongle.)
- Dongles that are registered with Release 24 (and on) will have the Reauthorization Expiry Date set to one year from the exact day of registration. For example, if you register the dongle on August 1, 2013, the Reauthorization Expiry Date will be August 1, 2014.

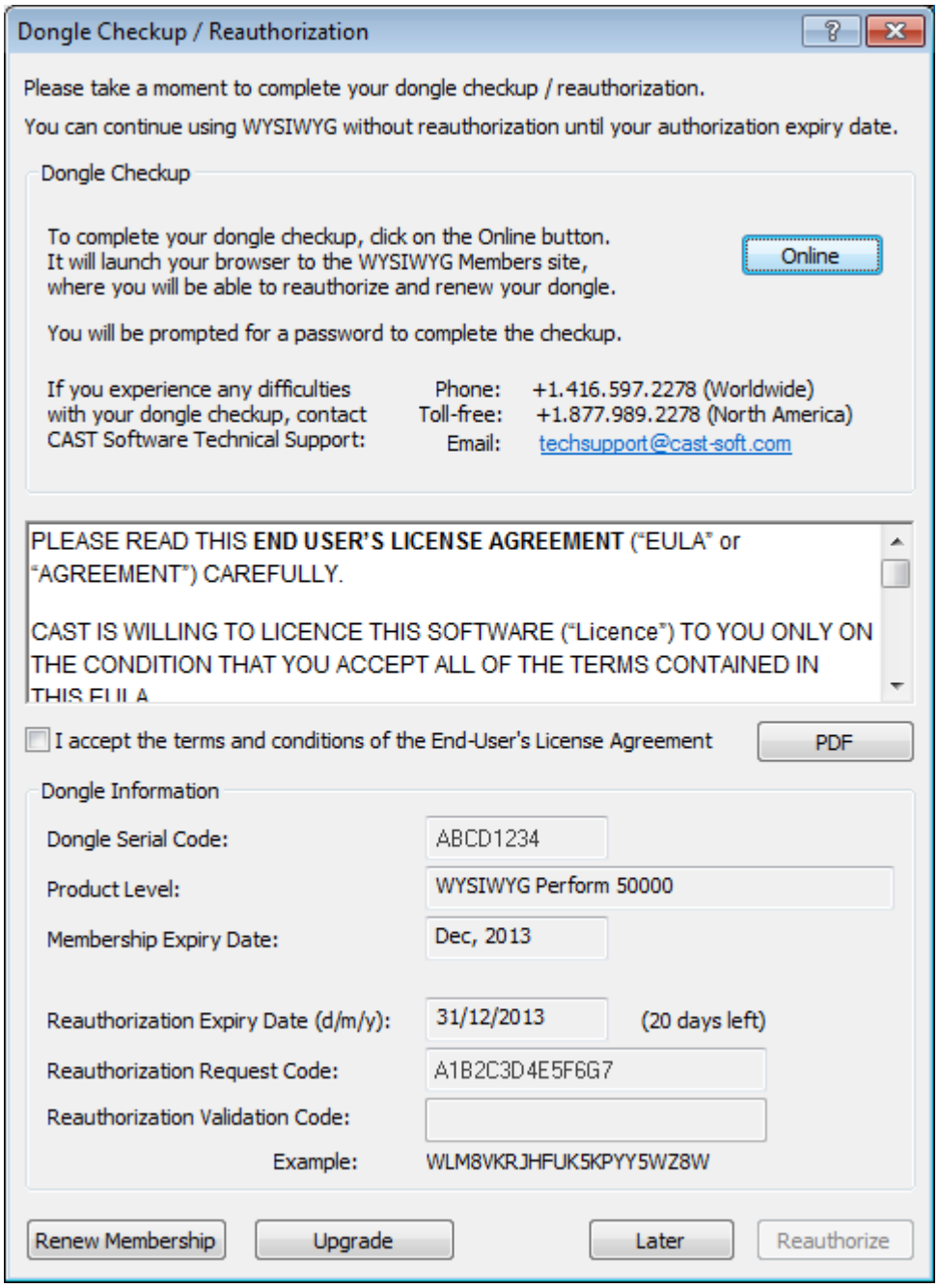
How it works

Exactly 30 days prior to the Reauthorization Expiry Date, regardless of when your Membership expires (or expired), every time you open WYSIWYG a window appears, requesting that you supply a Reauthorization Code for your dongle or the program will cease to function.

Note: You have 30 days from the first appearance of the Reauthorization window to complete the process. To postpone the process during this period and continue working in WYSIWYG, click *Later* in this window. At the 30-day mark, however, you *must* reauthorize your dongle or WYSIWYG will cease to function (until you complete the process).

To reauthorize your dongle

Note: You require an Internet connection to perform this procedure.



1 When the *Dongle Checkup/Reauthorization* window appears, click *Online* to open the WYSIWYG Members Only Area login page and begin the Reauthorization process.

2 Log into your account with your user name and password.

Note: If you do not have an account, you will be asked to create one by entering a password and your contact details. Ensure that the email address you provide is valid.

3 Edit any of your contact details, as required. Ensure that the email address you provide is valid. If you do not need to edit any information, or when finished, click *Submit*.

4 On the Annual Checkup page, click *Let's go*.

Result: The dongle's information appears, along with the options to Renew its Membership.

Note: If you do not wish to Renew the Membership at this time, deselect *Renew Membership*.

5 Click *Proceed* at the bottom of the page. (You may need to scroll down to see this button.)

Result: The Reauthorization page appears, displaying your Reauthorization Validation Code.

6 Double-click the code to select it and copy it by pressing CTRL +C.

7 Switch back to WYSIWYG.

8 Click the checkbox by "**I accept the terms and conditions of the End-User's License Agreement**" in the middle of the *Dongle Checkup/Reauthorization* window.

Result: The Reauthorization Validation Code field becomes enabled.

9 In the Reauthorization Validation Code field, paste the code (CTRL+V).

10 Click *Reauthorize*.

Result: A 'success' message appears.

11 Click *OK*.

Result: WYSIWYG launches. Your dongle is reauthorized.

Upgrades

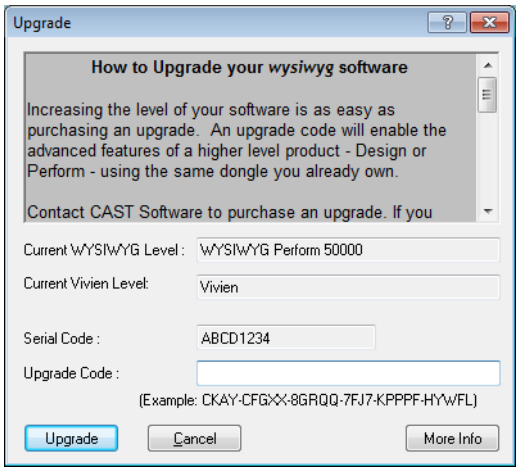
Introduction

An upgrade code enables you to run the advanced features of a higher level product, either Design or Perform, using the same dongle that you already own. Contact your local WYSIWYG dealer to purchase an upgrade. You will be given an upgrade code at the time of purchase.

To upgrade the level of your WYSIWYG product

1 Start WYSIWYG.

2 From the **Help** menu, choose **Upgrade**.



The current level of WYSIWYG installed on your computer is displayed in the *Current Level* box. The serial code of your dongle is shown in the *Serial Code* box.

3 In the *Upgrade Code* box, type your upgrade code, and then click *Upgrade*.

4 Exit WYSIWYG, and then restart your computer.

5 Launch WYSIWYG.

Result: The new level of software is displayed in the upper right corner of the WYSIWYG user interface.

Note: If you would like to purchase an upgrade but are unsure of who to contact, click *More Info* to obtain a list of WYSIWYG dealers.



Membership and Leases

Introduction

Starting with Release 24, you have two options for running WYSIWYG: you can purchase a Membership or you can lease a WYSIWYG dongle for one year.

Membership

When you purchase a Membership to WYSIWYG, you purchase the dongle. Your Membership is valid for a specific period of time, during which you can take advantage of all new software updates and releases. You can easily view and renew your Membership from within WYSIWYG by following the procedure “Obtaining Membership details” on page 40.

Leases

Starting with Release 24, you will be able to lease WYSIWYG dongles on a yearly basis. Leased dongles carry the same privileges as purchased ones (for example, software updates, library updates, and technical support). However, unlike purchased dongles, once the lease is over, WYSIWYG will no longer run with that dongle, giving you the choice of renewing the lease or purchasing the dongle.

Leasing a dongle compared to Membership

- If you choose to lease WYSIWYG for 12 months, you pay less than the cost of *purchasing* WYSIWYG, but more than the cost of *renewing* a WYSIWYG membership.
- With the Lease program, when the Lease Expiry Date is reached, the lease is over and the dongle will not run the software anymore (unless you choose to purchase another lease period or buy a WYSIWYG Membership).
- The Lease Expiry Date differs from the Dongle Reauthorization Date (associated with the Member program) in that once you reauthorize your dongle, you can continue to use WYSIWYG for another year. Further, if you choose not to purchase a Membership renewal but continue to reauthorize your dongle, you can continue to use your old (non-current) WYSIWYG software.
- A leased dongle can be re-leased at any time (for another year), at which point it will receive the latest version of WYSIWYG, plus all upgrades available that year.
- Unlike with Membership, a new Lease starts on the date when the Lease is purchased, not when the last Lease expired.
- At the end of your lease, you can convert to Membership. For pricing details, please contact Customer Service. You cannot (nor would it make sense to) switch from Membership to leasing. Instead, have the option to lease a new dongle as needed.

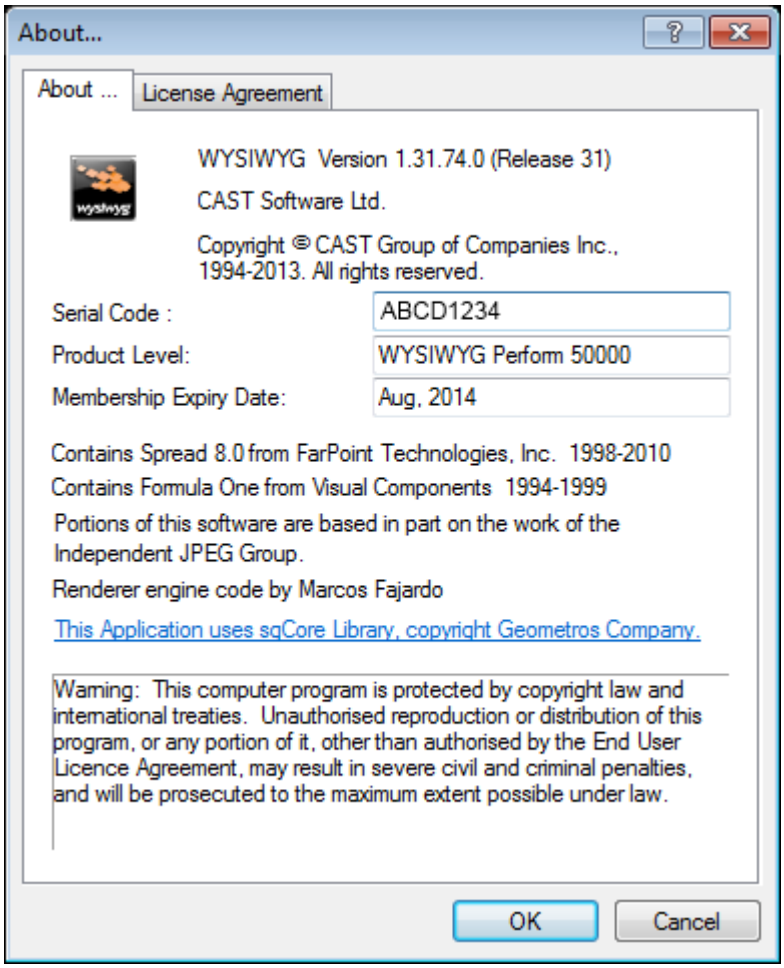
Obtaining Membership details

Details of your Membership are stored within WYSIWYG.

To obtain Membership details

From the **Help** menu, choose **About WYSIWYG.**

Result: The month and year that your Membership expires is displayed in the *Membership Expiry Date* box.



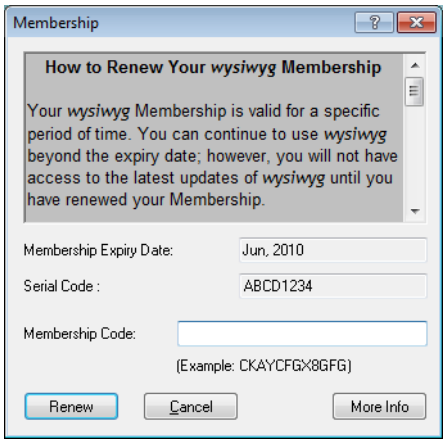
Renewing your Membership

Your WYSIWYG Membership is valid for a specific period of time. You can continue to use WYSIWYG beyond your Membership expiry date; however, you will not be able to access software updates until you renew your Membership.

Contact your local WYSIWYG dealer to purchase a renewal for your Membership. CAST Software provides you with the Membership code that you require.

To renew your Membership

1 From the **Help** menu, choose **Renew Membership.**



The date when your WYSIWYG Membership expires is displayed in the *Membership Expiry Date* box.

2 In the *Membership Code* box, type the code. The code is 10 alphanumeric characters long.

3 Verify that you have entered the correct information.

4 Click *Renew*.

5 Exit WYSIWYG, and then restart the software for the renewal to take effect.

Note: If you would like to renew your Membership but are unsure of who to contact, click *More Info* to obtain a list of WYSIWYG dealers.

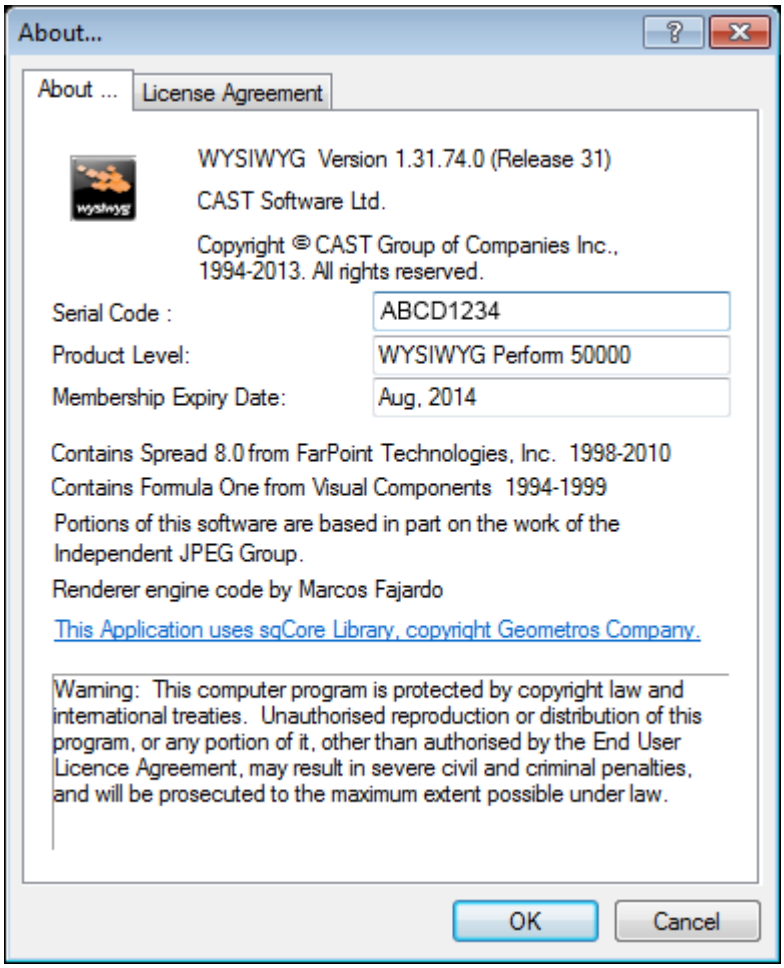
Obtaining Lease Details

Details of your Lease are stored within WYSIWYG.

To obtain Lease details

From the **Help** menu, choose **About WYSIWYG.**

Result: The date when your Lease expires is displayed in the *Lease Expiry Date* box.



Renewing Your Lease

Your WYSIWYG Lease is valid for one year from the date of purchase. You cannot continue to use WYSIWYG beyond your Lease expiry date; you must either renew your lease, or purchase a Membership to use the program.

Contact CAST Software or your local WYSIWYG dealer to purchase a renewal for your Lease.

To renew your Lease

Starting 30 days before your lease expires, the Dongle Lease Renewal window appears each time you open WYSIWYG until the Lease Expiry Date is reached, prompting you to renew your lease (or purchase a Membership).

1. In the Dongle Lease Renewal window, click the check box if you accept the terms of the license agreement.
2. Note the Lease Request Code shown in this window.
3. Call CAST Software Customer Service at one of the numbers provided.
4. When requested by the Customer Service Representative, recite the Lease Request Code.
5. The Customer Service Representative will give you the Lease Validation Code. Type the code in the box provided.

6 Click *Renew*.

Result: A Lease Renewal Successful message appears and your dongle is leased for another year.

7 Click *OK*.

Result: WYSIWYG starts.

Passwords

Introduction

You can set a password in WYSIWYG to password-protect your dongle.

To set a password

1 From the **Help** menu, choose **Password**.

Result: The *Change Password* dialog box appears.



2 In the *New password* box, type your new password. The password can be any combination of letters, numbers, and characters. As you type the password, WYSIWYG displays an asterisk (*) for each character that you type. The length of the password can be between 6 and 20 characters.

3 In the *Confirm password* box, re-enter the password.

4 Click *OK*.

Result: The next time that you launch WYSIWYG on the same computer, you will be prompted to enter your password. In the *Enter password* box, type your password. If you do not want to be prompted to enter a password every time you launch WYSIWYG, select the *Remember for next time* check box.

ATTENTION: If you forget the password that you set on the dongle, **the only way to reset if is by returning the dongle to the CAST Software office in Toronto, Canada**. **You will not be able to use WYSIWYG while the dongle is in transit.** As such, it is strongly recommended that you make note of the password for safekeeping.

To change a password

1 From the **Help** menu, choose **Password**.

Result: The *Change Password* dialog box is displayed.

2 In the *Current password* box, type the password that you previously set. You must type the current password if you want to change the password. As you type the password, WYSIWYG displays an asterisk (*) for each character you type.

3 In the *New password* box, type a new password.

4 In the *Confirm password* box, re-enter the password.

5 Click *OK*.

Note: If you decide that you do not want to password-protect your dongle, select the *No password* check box.

Chapter 4

WYSIWYG user interface



All of the WYSIWYG levels use the same set of screens. The user interface was designed to facilitate smooth transitions from mode to mode, and to keep the data organized and easy to find.

In this chapter

Navigation and screen set-up	49
WYSIWYG User Data Profile	53
Shortcut bars	55
Toolbars	59
Status bar	68
Wireframe views	71
Shaded views	78

Navigation and screen set-up

Welcome window

When you start WYSIWYG, the *Welcome* window appears. The application level is shown in the upper right corner of the window.



From the **File** menu, you can choose to open an existing file or create a new file. A list of template files are shown on the left and a list of recently saved files are shown on the right.

Click a file name to start working with WYSIWYG.

Modes

The mode buttons located along the top of the user interface are used to access the different working modes available within the WYSIWYG levels.



Click the appropriate mode button to change modes.

CAD mode

CAD mode is where you create your show drawings or “model.” This includes drawing your venue, set pieces, lighting positions, focus positions, and lighting fixtures. This is also where you can import CAD files. CAD mode operates like many other CAD programs, so many of the concepts will be familiar to those who have used a computer-

aided drafting program before. WYSIWYG adds features that are specific to the entertainment industry, such as a comprehensive 3D library containing truss, lighting equipment and accessories, as well as props, musical instruments, and various human figures.

In WYSIWYG Report, you are limited to 2D views of your plot. In Design and Perform, 3D isometric and shaded views are available.

Data mode

As you draw in CAD mode, all of your fixture information is compiled into spreadsheets that you can access in Data mode. In this mode, you can view and edit the data for all your fixtures. Data mode operates like many other spreadsheet programs, so many of the concepts will be familiar to those who have used a spreadsheet program. The CAD and Data modes remain completely interactive and any information modified in either mode is updated in the other.

Design mode

Design mode was created to provide lighting designers with an avenue for experimentation. Design mode can be used as a troubleshooting tool or as a creative tool to help you come up with cue concepts. In Design mode, you can create static lighting looks using the design tools, and then save and render those looks to output photo-realistic pictures. You can turn on and control fixtures without having to patch or connect to a console.

Presentation mode

Presentation (Pres) mode contains all the tools necessary for creating professional printouts of your show document including reports, plots, and images. As with all other modes, the contents of the Pres mode are continually updated as you draw, input data, and modify your show file. In addition to being able to create your own plots, WYSWIYG provides a series of default plots and reports that are ready for printout. These defaults can be used as is or customized to suit your preferences.

Live mode

Live mode is used for graphically simulating the output of a lighting control console or compatible offline editor. This is where you can pre-cue and visualize your show. Live mode also has rendering ability so that you can create photo-realistic pictures of the simulated lighting looks. Live mode will fully display a console's output, including transitions from cue to cue, allowing you to see the programmed changes to lights over time.

Availability of modes

The following working modes are available in WYSIWYG Report:

- CAD 2D
- DATA
- PRES 2D

The following working modes are available in WYSIWYG Design:

- CAD
- DATA
- DESIGN
- PRES

The following working modes are available in WYSIWYG Perform:

- CAD
- DATA
- DESIGN
- PRES
- LIVE

Layouts

Beneath the work area in each mode is a series of layout tabs. A layout is a configuration of work views. Views and layouts are mode specific. The layouts are not editable and there are no options for saving custom layouts. To this end, WYSIWYG supports popup frames and has multi-monitor functionality, thus providing alternative tools for customizing your work environment. These features are discussed below.

Click the appropriate layout tab to change layouts.

Views

Views are the windows in the work space. The windows are pre-configured on the screen based on the layout, as discussed above.

The WYSIWYG views include: wireframe, wireframe-selection-only, flight case, shaded, spreadsheet, patch, error, report, image, plot, and worksheet.

The layouts and views available in each mode are discussed at the beginning of each chapter.

For more information, see “Wireframe views” on page 71 and “Shaded views” on page 78.

Popup frames

You can place a view in a “popup frame” so that it remains active across modes. For example, if you place the patch view in a popup frame, it will remain visible even if the mode is changed from Data to CAD.

The position of popup frames are remembered when a file is saved. When the saved file is reopened, all popup frame positions will be remembered and set exactly how they were working last.

To place a view in a popup frame

1 Make the desired window active.

2 From the **Options** menu, choose **Open in Popup Frame**.

Tip: Alternatively, click the *Frame Popup* tool on the *View* toolbar. 

Result: A new window containing a copy of the selected view opens.

Popup frames

- retain all the functionality of the original view so you can work in the popup view if you want to
- are fully integrated with the remaining show document and are continually updated (and vice versa)
- can be resized and moved to any location on the screen or across monitors

To display popup frames in full screen mode

1 Make the popup window active.

2 Click the *Full Screen* button. 

Result: The popup is expanded to fill the entire screen with no title bar visible.

Tip: To return the popup frame to normal size, press *ESC*.

To minimize a popup frame

Double-click the title bar of the popup window.

Result: The title bar only is displayed.

Tip: You can also click the *Minimize* button. 

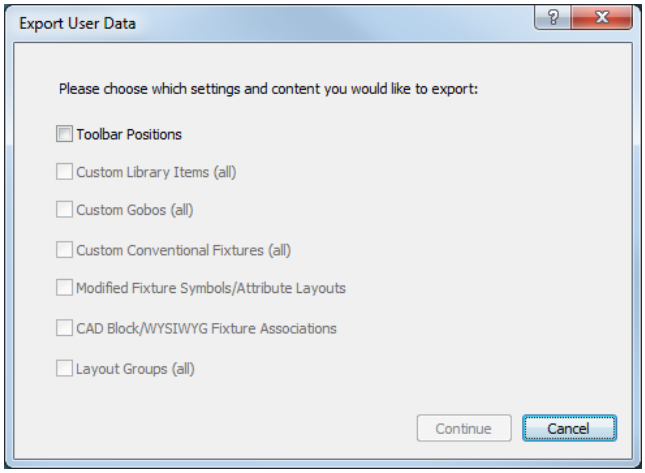
WYSIWYG User Data Profile

Introduction

You can save WYSIWYG settings that are specific to a particular installation of the WYSIWYG software into an external file with a .wud extension. You can then load this file into WYSIWYG R26 and beyond to automatically load all the user settings saved in the .wud file. With .wud files, you can share your custom user data between multiple computers and when upgrading to newer WYSIWYG releases.

Exporting User Data

WYSIWYG User Data files can only be imported and exported from the WYSIWYG Welcome window. To save your existing settings, click **File** > **Export User Data**. A Wizard guides you through the export process.



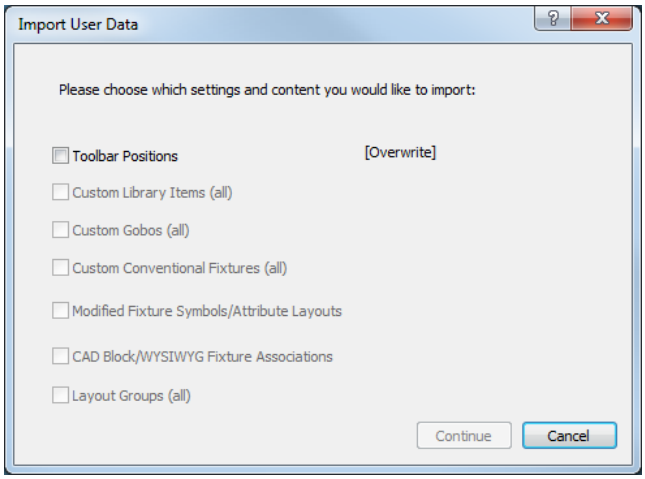
You can save the following user settings in a .wud file:

- Modified Fixture Symbols/Attribute Layouts
- Custom Conventional Fixtures
- Custom Gobos
- Custom Library Items
- CAD Block /WISWYG Fixture Associations
- Toolbar Positions

Importing User Data

WYSIWYG User Data files can only be imported and exported from the WYSIWYG Welcome window. To load user settings from an existing .wud file, from the Welcome Screen, click **File** > **Import User Data**. A window prompts you to select the .wud file to open.

Once you select the file, a new window appears:



Use the checkboxes to select the appropriate settings. Note that an [Overwrite] text item appears to notify you when the existing setting will be overwritten with the settings stored in the *.wud file. Once the process is complete, you will need to restart WYSIWYG to load the new settings.

Shortcut bars

Introduction

Shortcuts are created for quick access to library items and view types, among other things. The shortcut bars are located on the left side of your screen. Shortcut bars are mode and layout specific. Each layout has designated shortcut bars.



In general, there are three types of shortcuts:

- navigation
- library
- views/tools

Navigation shortcuts

The first type of shortcut is used for navigation purposes. The navigation shortcuts are:

-  Navigation
-  Views

Navigation shortcuts store the mode and layout destination. For example, you can save a navigation shortcut to Data mode / Patch layout. The next time you want to go back to that location, click the shortcut to save you at least one step.

View shortcuts store the plot type and zoom level. View shortcuts only apply to wireframe views.

Library shortcuts

The second type of shortcut is used in place of browsing the library. The library shortcuts are:

-  Fixtures
-  Accessories
-  Color
-  Gobo
-  Library
-  Truss

Views/tools shortcuts

The final type of shortcut is used to open and save views and tools. The views/tools shortcuts are:

-  Columns
-  Patch
-  Errors
-  Worksheets
-  Images
-  Layouts
-  New Plots
-  Legend
-  Key
-  Report
-  Looks
-  Fixture Groups
-  Palettes
-  Render

Working with shortcuts

Generally the procedure for creating, modifying, and working with shortcuts is the same regardless of the type of shortcut.

To create a new shortcut

1 Right-click in open space on the appropriate shortcut bar, and then choose the appropriate **New option**.

Note: For navigation type shortcuts, ensure that the destination is set up, and then proceed with step 1.

2 Based on the type of shortcut, you are prompted to select the object or type a name for the new shortcut.

3 Click *Insert* or *OK*.

Result: The shortcut is created and added to the bottom of the list on the shortcut bar that you selected in step 1.

To remove a shortcut

1 Right-click the shortcut icon that you want to remove.

2 Choose **Delete**.

Result: You are prompted to confirm the deletion.

3 Click *OK* to confirm.

Result: The selected shortcut is deleted.

To rename a shortcut

1 Right-click the shortcut icon that you want to rename.

2 Choose **Rename**.

3 Type a new name for the shortcut.

4 Click OK.

Result: The selected shortcut is renamed.

To clone a shortcut

1 Right-click the shortcut icon that you want to clone.

2 Choose **Clone Shortcut**.

3 Type a name for the copy.

Result: The selected shortcut is cloned and the copy is added to the bottom of the list.

To arrange shortcuts

You can arrange shortcuts in the shortcut bar area by using drag-and-drop functionality.

1 Click the shortcut that you want to move.

2 While holding down the mouse button, press ALT to move the shortcut to the desired location.

3 Release the mouse button to place the shortcut.

To set the display size for shortcuts

Right-click in open space on the appropriate shortcut bar, and then choose either **Icons** or **List** (depending on your current setup).

Result: When you choose **Icons**, all shortcuts are displayed as icons. **List** will cause the shortcuts to be listed in the shortcut bar area, thereby allowing you to fit more shortcuts in the area.

The settings are retained for all shortcuts.

Note: To scroll to the bottom of a long list of shortcuts, use either the scroll button on your mouse or the scroll bar along the side of the shortcut bar.

To use a shortcut

Click the shortcut icon.

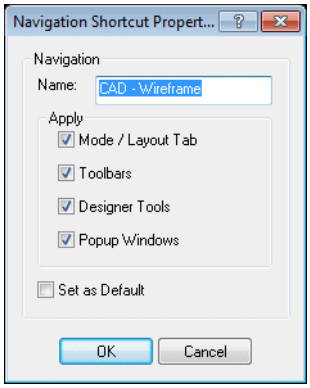
As with all objects in WYSIWYG, all shortcuts have properties that affect the object being inserted, the view or tool, or the navigation method.

To modify a shortcut's properties

1 Right-click the shortcut icon that you want to modify.

2 Choose **Properties**.

Result: The properties dialog box is displayed. The settings modified here affect the object, view, tool, or navigation destination of the selected shortcut. An example of the *Navigation Shortcut Properties* dialog box is shown below; however, the shortcut properties window varies based on the type of shortcut that you select.



3 In the *Name* box, type the new name of the shortcut.

4 To open the shortcut in a pop-up frame, click the *Pop-Up Window* check box. For more information on popup frames, refer to "Popup frames" on page 51.

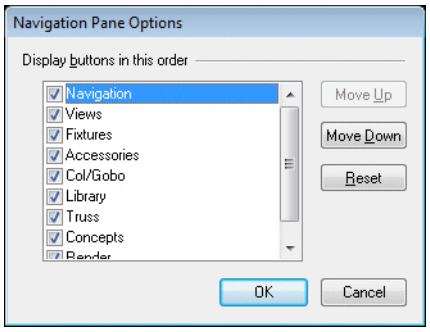
To configure the shortcut bar

You can configure the shortcut bar to show fewer/more categories, and reorder the categories that you see.

1 In the bottom right corner of the shortcut bar, click the *Toolbar Options* arrow.

2 In the resulting pop-up menu, you have the following choices:

- *Show More Buttons* Click to add the next shortcut category to the bottom of the shortcut bar. This button is enabled only if you have hidden some shortcut categories.
- *Show Fewer Buttons*: Click to remove the bottom-most shortcut category from the shortcut bar. Click consecutively to remove each category shown from the bottom upwards.
- *Navigation Pane Options*: Click this button to arrange the order of the shortcut categories shown in the shortcut bar.





Toolbars

The menu bar

The menu bar is located within the WYSIWYG user interface, below the mode buttons and above the work area. The available menus change when moving from mode to mode. The individual menus are explained within the context of each mode chapter.

The menu bar is dockable. It has a grab bar on the left that is used to move the menu bar around the screen. It may be placed on the edges of the work area; top, bottom, left or right. It may also be dragged off the edge of the work area and into its own window. This window remains on top of the WYSIWYG screen and may be dragged anywhere on your display.

Menu commands can be accessed using a mouse, keyboard, or by using hot keys.

To access menu commands using the keyboard

1 Press ALT+*n*, where *n* is the underlined letter in the menu name.

Result: The menu will be displayed.

2 Press the key corresponding to the underlined letter in the command that you want to execute.

Note: Some commands have shortcuts that do not require menu selection. In those cases, the shortcut keys are listed to the right of the command in the menu.

Example:
To undo the last command, press CTRL+Z.

Toolbars

Toolbars provide button access to most commands. This is in lieu of selecting the commands through the menus in the menu bar.

Toolbars, like menus, are mode sensitive. However, unlike menus, you can customize how the toolbars are displayed and which toolbars are open for each mode. Toolbars that are greyed out in the toolbar selection list are not used in the current mode.

Toolbars are typically found directly under the menu bar and have a grab bar on the left that is used to move the toolbar around the screen. They can be placed on the edges of the work area (top, bottom, left or right), and can also be dragged off the edge of the work area and into their own window. This window remains on top of the WYSIWYG screen and can be dragged anywhere on your display.

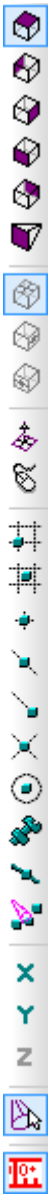
The following toolbars are available (listed in alphabetical order).

Align toolbar (CAD, PRES/Layouts and /New Plots)



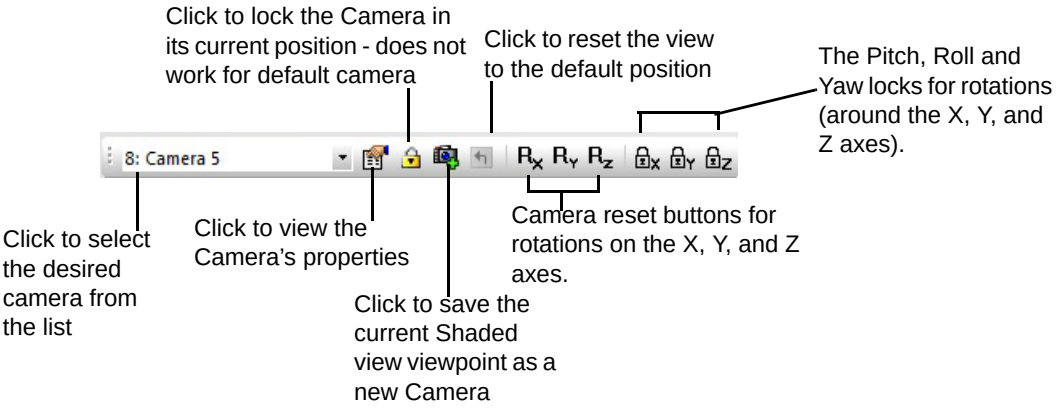
Align to first selected
Align to last selected
Align to extents
Align top
Align bottom
Align left
Align right
Center horizontally
Center vertically

CAD Options toolbar



Plan
Left
Right
Front
Back
Isometric
WorkPlane Plan
WorkPlane Side
WorkPlane Front
Missing coordinate
Interactive
Grid Snap
Mid-Grid Snap
Point Snap
Midpoint Snap
Endpoint Snap
Intersection Snap
Center Snap
Assembly
Pipe
Focus
Ortho X
Ortho Y
Ortho Z
Toggle Beam Dragging
Ruler Toggle

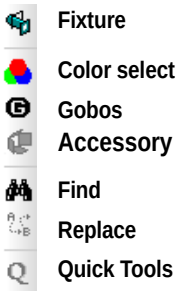
Camera toolbar



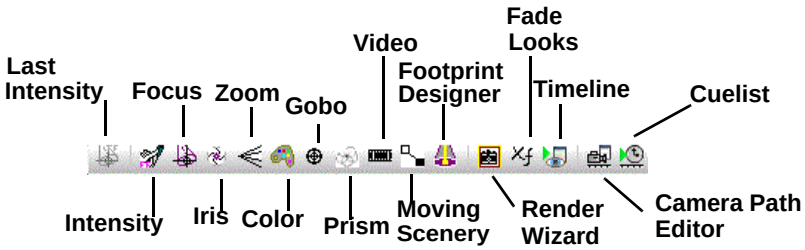
ComEdit toolbar



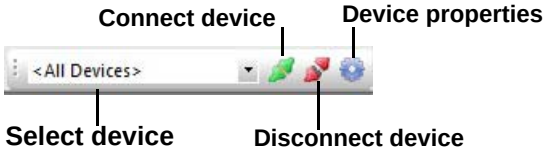
Data toolbar



Design toolbar



Device Manager toolbar



Draw toolbar



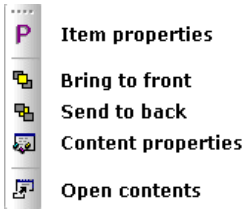
Point
Solid line
Center line
Hidden line
Dotted line
Spline
Circle
Arc
Elliptical Arc
Rectangle
Polygon
Text label
Dimension
Surface
Riser
Cylinder
Cone
Sphere
Wall
Library
Pipe
Curved Pipe
Truss
Fixture
Accessory
Focus position
Focus line
Focus arc
LED Wizard
Pipe and Drape Wizard

Edit toolbar

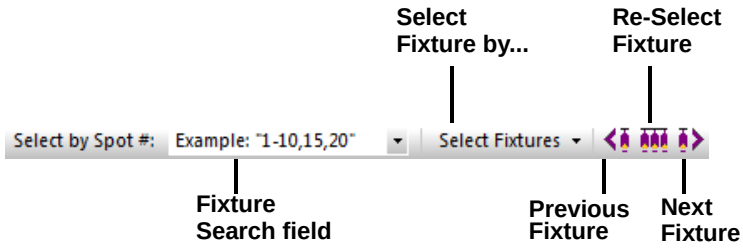


Item properties
Cut
Copy
Paste
Delete
Undo
Redo
Repeat last command

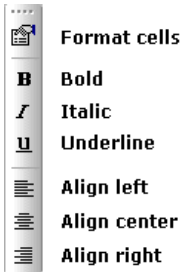
Edit (Pres) toolbar



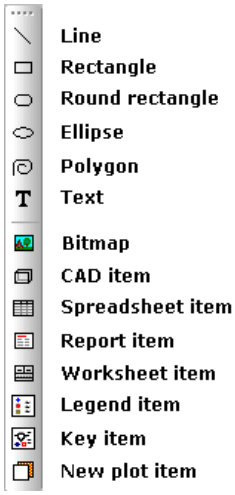
Fixture Section toolbar



Format toolbar



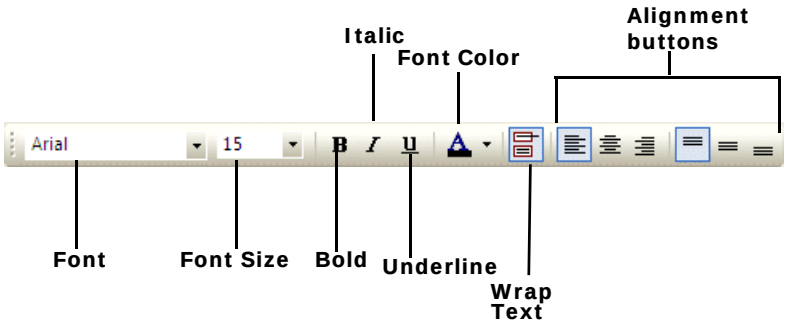
Insert (Pres) toolbar



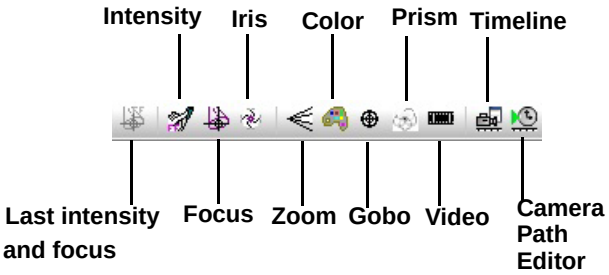
Layers toolbar



Layout Pres Text toolbar (Pres > Reports, Layouts and Worksheets)



Live toolbar



New Plots Options



New Plots View

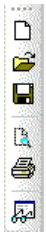


Scenes toolbar



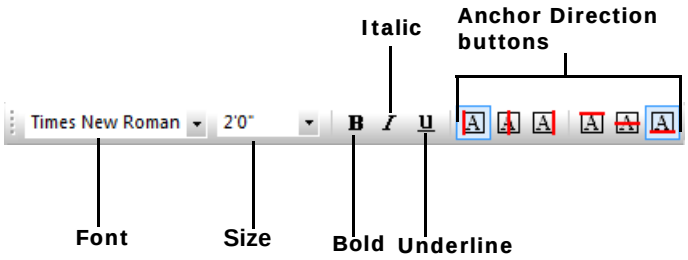
Current scene

Standard toolbar

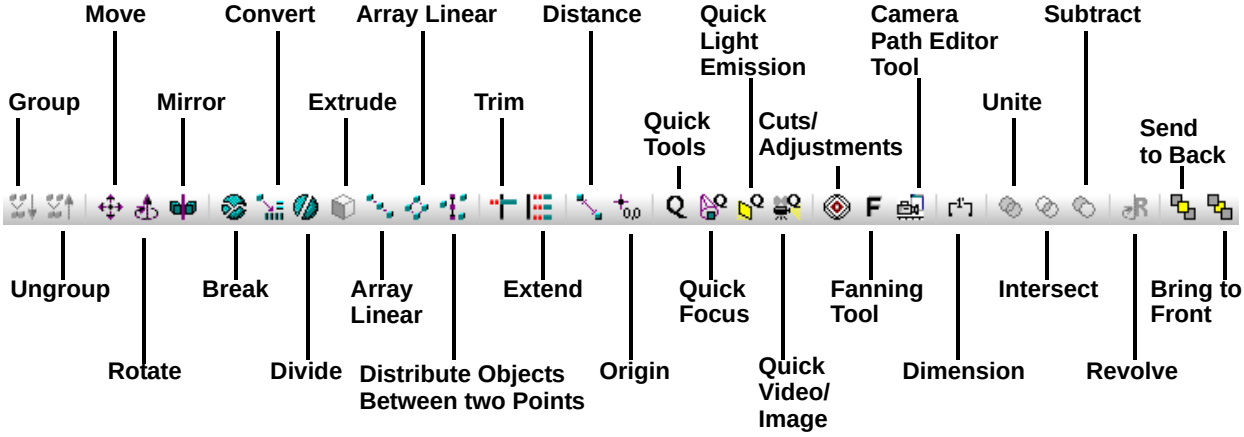


New document
Open
Save
Print preview
Print
View options

Text toolbar (CAD, and Pres > New Plots)



Tools toolbar



View toolbar



To hide and display toolbars using the toolbar list

1 Right-click in the toolbar area.

Result: The toolbar list is displayed.

2 Click on the name of the toolbar that you wish to display. A check mark to the left of the toolbar name indicates it is currently displayed.

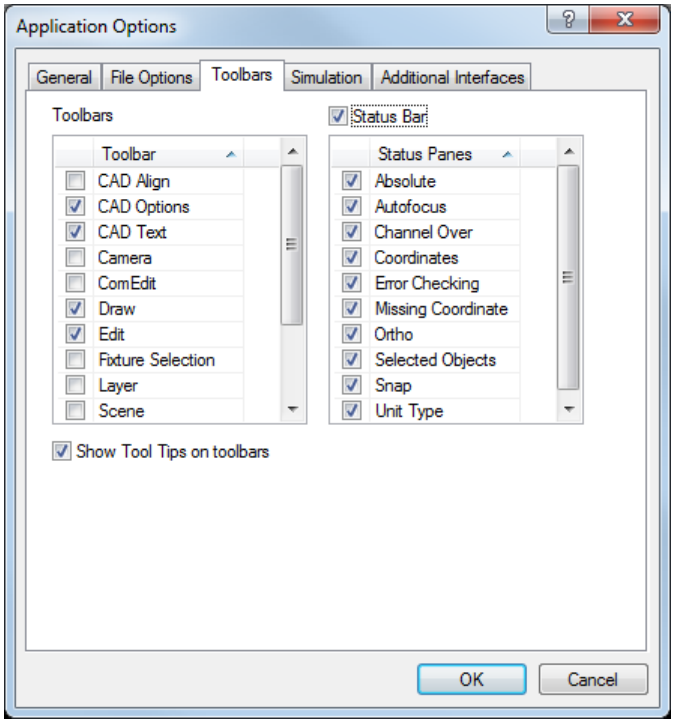
3 Repeat to display or hide multiple toolbars.

Tip: From the toolbar list, select *Toolbars* to open the *Application Options* dialog box. In this box, you can choose multiple toolbars and configure other screen options, such as the status bar and tooltips.

To hide and display toolbars using the Options menu

1 From the **Options** menu, choose **Application Options**.

2 Click the *Toolbars* tab.



3 Select the check boxes of the toolbars that you want to display.

Note: Since toolbars are mode-sensitive, certain toolbars are disabled based on the mode that you are currently in.

Floating Toolbars

Floating Toolbars are available in WYSIWYG to offer common features conveniently at your cursor. When you right-click on a view, in addition to the usual right-click menu, a floating toolbar will appear either above or below the right-click menu (depending on where you clicked on the screen).

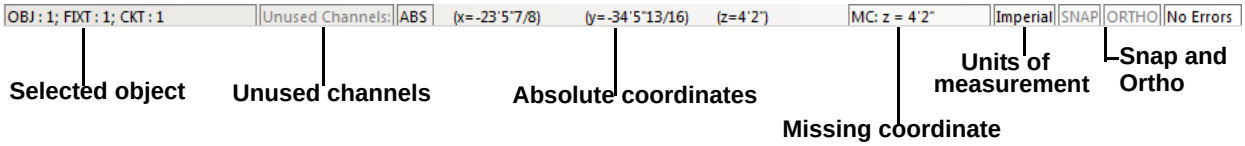


The icons displayed are the same as the ones displayed in the menus and toolbars. If you are not familiar with an icon, hover over it and a tooltip will appear displaying the feature's name.

Status bar

Introduction

The status bar is displayed along the bottom of the WYSIWYG screen, below the layout tabs. The status bar displays the prompt line, the number of unused channels (WYSIWYG Perform), selected object information, and tracks the status of the snap, ortho and absolute coordinates commands. It also tracks and displays the coordinates of the cursor.



The status bar can be turned on or off from the toolbar list. You can also customize the contents of the status bar from the Application Options window.

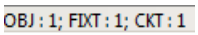
For more information on setting the options of your drawing, refer to “Application Options window” on page 389.

Prompt line

The prompt line displays the current status of a command within your drawing. If you are currently working with a command that requires multiple steps (the placement of a pipe, for example), the prompt line displays a message indicating the next step required to accomplish that task. The prompt line also displays a short description of a command when you point to it using your mouse.

Selected object count

In this display, “O” stands for the number of objects currently selected. The “F” stands for the number of fixtures selected and the “C” stands for the number of circuits currently selected.



Unused channels

Although there is never a limitation on how many fixtures can be patched in a file, there is a limitation on the number of DMX channels that can be simulated in Live mode. WYSIWYG will not simulate the DMX values being received on channels exceeding the Perform channel count of your system. WYSIWYG Perform is available in 500, 1000, 2000, 5000 and Unlimited channels. (WYSIWYG Unlimited has 50,000 channels.) The unused channels displayed on the status bar track the remaining channels for simulation based on the level of your Perform system. Note that fixtures on layers that are not visible, or which are not included in the current scene, are not counted.

Absolute and Relative coordinates

This setting determines how coordinate information is displayed mid-command. In ABS (Absolute) mode, which is the default, the coordinates display always shows the coordinates currently under your cursor. In REL (Relative) mode, the Coordinates display shows the relative distance and angle from the last position, indicated with a mouse click.

Coordinates

Coordinates are displayed in the order of X, Y and Z. The coordinates display the position of the cursor as a distance from the origin of the drawing. For more information on coordinates and setting the origin point, refer to “Coordinate system and origin” on page 98.

Missing coordinate

Double-click this label to set the missing coordinate. The missing coordinate (X,Y,or Z) is the coordinate whose value cannot be entered by clicking on the screen. The easiest way to determine the missing coordinate in a wireframe view is to move the mouse around and look at the status bar at the bottom of the working area. You will see only two values changing. The value that is not changing is the missing coordinate for that view or workplane.

The missing coordinate is dependent on the plot type and the workplane selected. Once entered this value will affect all subsequent objects inserted in the current view. For example, if the missing coordinate is set to 5’ in a plan view, all objects will be placed 5’ off the floor (X,Y,5) until the missing coordinate is changed again. Note that the missing coordinate is not a move tool; objects are not moved to the missing coordinate value. The missing coordinate only affects subsequent inserts.

Units of measurement

Click this label to toggle between metric and imperial units of measurement.

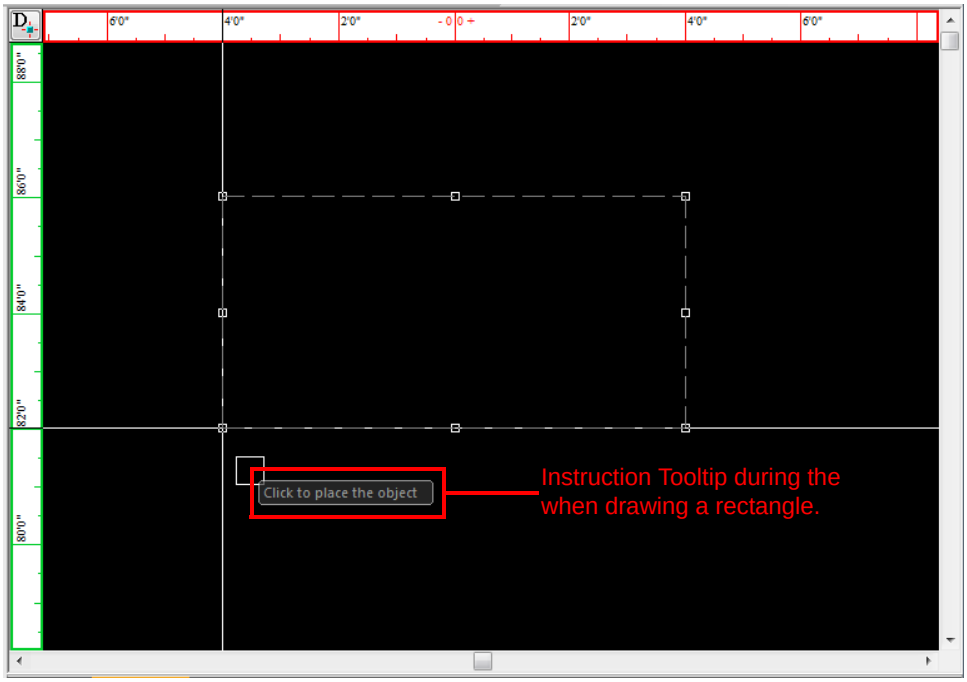
Snap and Ortho

The snap and ortho entry shows the status of these items. If the indicator is black, a snap or ortho setting is active. If the indicator is grey, snap and ortho are not active.

Instruction Tooltip

The Instruction Tooltip is a helpful window that appears automatically when you start an operation. The Instruction Tooltip explains how to perform the current operation. The information shown is the same information that is displayed in the Status Bar.

Note: By default the Instruction Tooltip is turned on.



To turn off Instruction Tooltips

1 From the **Options** menu, choose **Application Option...** .

2 In the **General** tab, unmark the **Enable Instruction Tooltip** checkbox.

Wireframe views

Introduction

When you first create a new show document and enter CAD mode, WYSIWYG defaults to the Wireframe view. All drawing is done in Wireframe view. Click the Quad tab to display three Wireframe views and a shaded view of your drawing.

Keyboard and mouse control

To modify the point of view

- The arrow keys move you in their respective directions.
- The PAGE UP and PAGE DOWN keys zoom you in and out. You can also roll the middle mouse button up and down to zoom in and out. The zoom action is centered on the mouse pointer rather than the center of the window.
- In a 3D perspective view (isometric or shaded), the CTRL key, in combination with any of the aforementioned keys, rotates you around your drawing.
- Holding down the middle mouse button will allow you to drag the drawing around the window. Alternatively, the *Pan* tool on the *View* toolbar also enables this type of movement  .
- At any time you can use the SHIFT key in combination with any of the movement keys to move in smaller increments.
- In shaded views, the mouse can substitute all key strokes. Click and drag to pan around and use the mouse wheel for zooming.

Zoom tools

Zoom tools allow you to view smaller or larger sections of a plot or drawing. There are seven zoom tools available in WYSIWYG.

Note: The zoom tools are not applicable in the Shaded view; to zoom in or out in this view, use the arrow keys or the roller wheel on your mouse instead.

To access the Zoom tools

1 From the **Zoom** menu, choose one of the Zoom tools.

Tip: You may also use the Zoom tools on the *View* toolbar.

Result: The viewpoint is adjusted accordingly.

The Zoom tools that are available are listed in the following table.

Zoom tool		Description
Zoom In		Moves your viewpoint closer to the center of the view.
Zoom Out		Moves your viewpoint farther away from the center of the view.

Zoom tool	Description
Zoom Fit 	Adjusts the viewpoint so that the extremities of the drawing fit into the current view.
Zoom Fit All (only available on the Zoom menu)	For Quad layout, adjusts the viewpoint so that the extremities of the drawing fit into the three Wireframe views simultaneously. In Wireframe view, adjusts the viewpoint so that the extremities of the drawing fit the window.
Zoom Previous 	Resets the view to the previous pan and zoom settings.
Zoom Next 	If you used the Zoom Previous tool, Zoom Next sets the view to the next pan and zoom settings.
Zoom Window 	Allows you to specify the area of the drawing to be viewed. For more information on using this tool, refer to the procedure below.

To use the Zoom Window tool

1. In Wireframe view, from the **Zoom** menu, choose the **Zoom Window** tool.
2. Click the left mouse button and drag a window around the area into which you want to zoom.
3. Click the left mouse button again to capture the second point of the window.

   Result: The view changes to the area that you have selected.

View shortcuts

View shortcuts record the plot type as well as the zoom and pan settings of a specific point of view. You can use the shortcut to get back to that exact view point the next time you need to.

To record a view shortcut

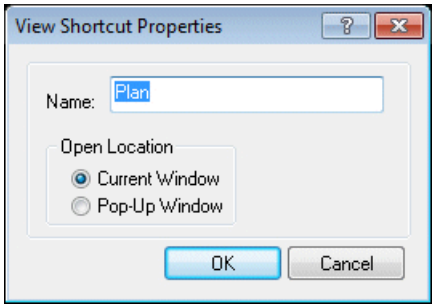
1. Set up the active window for the plot type, zoom, and pan settings you wish to record.
2. Right-click on the *Views* shortcut bar, and then choose **New View**.
3. Type a name for the new shortcut, and then click *OK*.

   Result: The shortcut is recorded and available on the *Views* shortcut bar.

To modify the properties of a view shortcut

1 Right-click on the Views shortcut icon that you want to modify, and then select **Properties**.

Result: The *View Shortcut Properties* dialog box is displayed.



2 In the *Name* box, type the new name of the shortcut.

3 To open the shortcut in the current active window, enable the *Current Window* option button.

4 To open the shortcut in a pop-up frame, enable the *Pop-Up Window* option button. For more information on popup frames, refer to “Popup frames” on page 51.

Modifying wireframe views

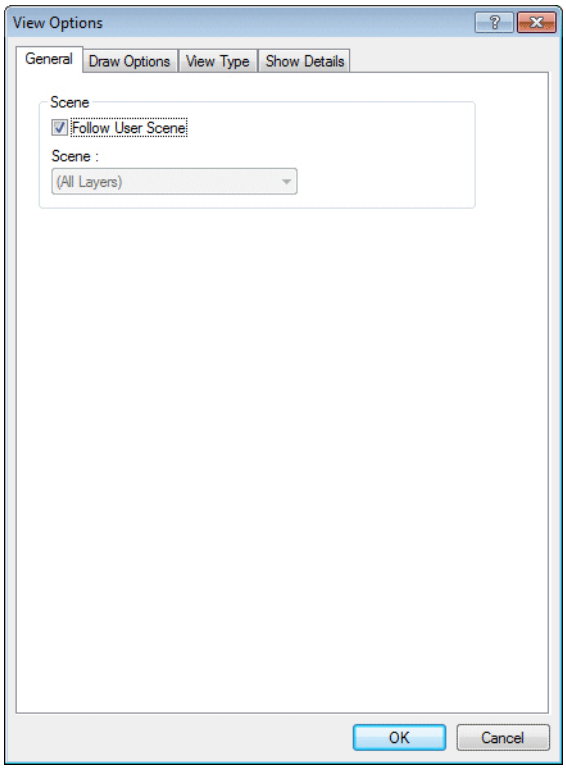
The properties of a wireframe view affect how objects are drawn and how much information is visible. These settings are modified in the View Options.

To modify a wireframe view

1 Ensure the wireframe view that you want to modify is active.

2 From the **Options** menu, choose **View Options**.

Result: The *View Options* window appears.



Tip: You can also use the *View Options* tool  on the *Standard* toolbar to open the view options.

3 Modify options as desired.

4 Click *OK*.

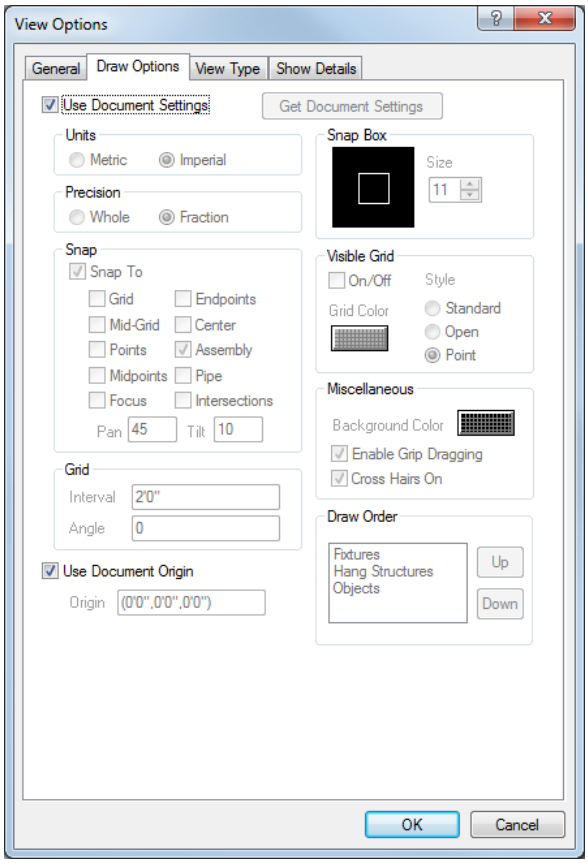
General tab

Options on the *General* tab affect the current scene.

- *Name*: Name of the wireframe view.
- *Follow User Scene*: Select this check box to use the currently selected scene. Click to clear this check box, and then select the desired scene from the *Scenes* drop-down list. To learn more about scenes, refer to “Editing objects” on page 185.
  Tip: You can also change the current scene at any time by using the drop-down list on the *Scene* toolbar.

Draw Options tab

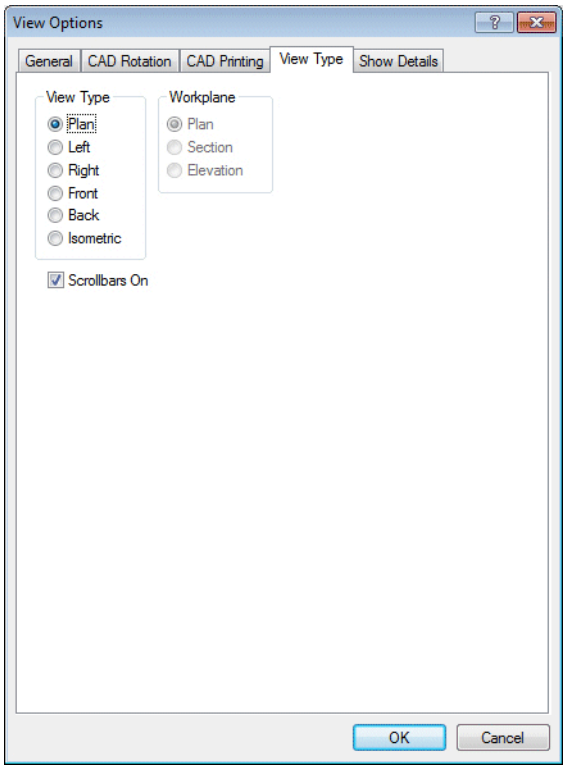
Options on the *Draw Options* tab affect the draw defaults. This tab is similar in function to the *Draw Defaults* tab available in User Options. For more information on the *Draw Options* tab, refer to “Draw Defaults tab” on page 404.



- *Use Document Settings*: Select this check box to reference the options selected on the *Draw Defaults* tab of *User Options*. Clear the *Use Document Settings* check box to make specific changes for the active view.
- *Get Document Settings:* Retrieves the values set on the *Draw Defaults* tab of *User Options*.

View Type tab

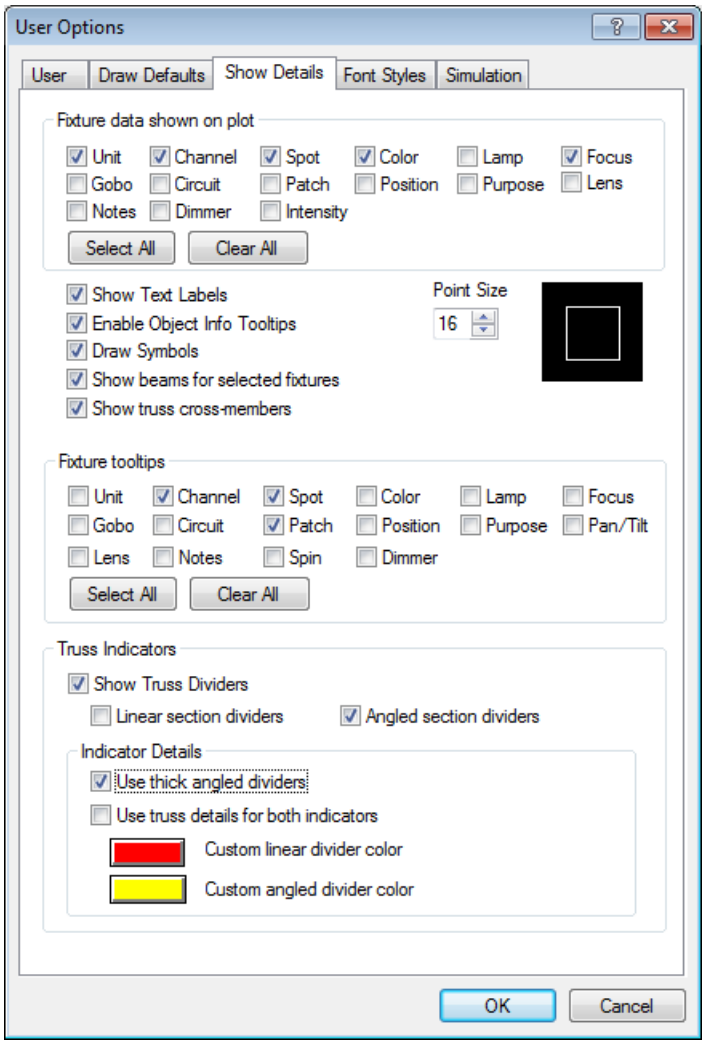
Options on the *View Type* tab affect the plot type of the active view.



- *View Type*: The view types available are as discussed in “Plot types” on page 104. **Tip:** The view type can be toggled using the tools on the *CAD Options* toolbar.
- *Scrollbars On*: Select this check box to display the scrollbars on the bottom and left side edges of the wireframe view. Click to clear this check box to turn the scrollbars off.

Show Details tab

Options on the *Show Details* tab affect what and how fixture, truss, text, and point information is displayed. This tab is similar in function to the *Show Details* tab available in User Options. For more information on the Show Details tab, refer to “Show Details tab” on page 405.



- *Use User Details*: Select this check box to reference the options selected on the *Show Details tab* of *User Options*. Click to clear the *Use User Defaults* check box to make specific changes for the active view.

Shaded views

Introduction

Shaded or OpenGL views offer a 3D perspective view of your drawing. Surfaces, truss, and fixtures are “filled-in” or solid, rather than outlined as in wireframe views.

- Shaded views are not available in WYSIWYG Report.
- Shaded views will follow Scenes just like any other view.

Keyboard and mouse control

To modify the point of view

- Use the arrow keys to move the position of the virtual target. PAGE UP/PAGE DOWN will zoom you in and out.
- At any time you can use the SHIFT key in combination with any of the movement keys to move in smaller increments.
- The mouse pan tool can be set up to rotate the model or the camera around the target. You can also use the mouse wheel to zoom in and out. The arrow keys will continue to move the target position as you do this.

To open Shaded view full screen mode

- You can expand the Shaded view to encompass the entire screen by right-clicking anywhere within the view and selecting **Open in Full Screen**. To exit full screen mode, press the ESC key.

Camera controls

Note: The Camera-to-Target distance is maintained for all operations listed below. To change this distance, hold down simultaneously CTRL and ALT and spin the mouse wheel up or down while the Shaded view or the shaded pane of Quad view is active. You need to perform this action to alter the Camera’s orbit; it also serves to set the target distance when rendering with Aperture enabled. This does not apply to Flying Cameras, since the target system for Flying Cameras is different than World or Standard Cameras’ target systems.

- Holding down the left Shift key on the keyboard while performing any of the actions below will decrease the speed at which changes are applied, regardless of how the actions are performed (i.e., by mouse or keyboard).
- Holding down the right Shift key on the keyboard while performing any of the actions below will increase the speed at which changes are applied, again regardless of how the actions are performed (i.e., by mouse or keyboard).

Orbit

Orbiting the camera causes it to spin around its target without altering the target’s position. Naturally, when the camera is orbiting, its own position changes. This control applies to World and Standard Cameras.

- **To orbit a Camera using the mouse**, click and drag with the left mouse button in the Shaded view or the shaded pane of Quad view.

- **To orbit a Camera by using the keyboard**, hold down ALT and tap the arrow keys while the Shaded view or the shaded pane of Quad view is active.

Notes:

- In order to prevent accidentally changing the shape of Camera Paths, it is not possible to orbit Flying Cameras.
- This is the "default" behaviour for a Camera in WYSIWYG; it has not changed despite the introduction of the new camera system in Release 29.

Pivot

Pivoting the Camera causes it to change its orientation/where it is pointing to (without altering its own position). Another way to think of it is that by pivoting the Camera, its target's position is altered.

This control applies to World, Standard and Flying Cameras (at user-defined nodes).

- **To pivot a Camera using the mouse**, hold down ALT on the keyboard and click and drag with the left mouse button in the Shaded view or the shaded pane of Quad view.
- **To pivot a Camera using the keyboard**, tap the arrow keys (without holding down ALT) while the Shaded view or the shaded pane of Quad view is active.

Notes:

- This is one of the two ways in which a Flying Camera can be modified at a use-defined node—dollying is the other.
- Flying Cameras cannot be pivoted at nodes where Behaviour was set to *Follow Path* or *Follow Target*. In cases where something is "interfering with the shot" and the Flying Camera at such a node must be pivoted, simply change the Behaviour at that node to user defined—the Camera's current position and orientation will not be altered by the Behaviour change.

Dolly

Dollying the Camera causes its own position and its target's position to be altered at the same time. This control applies to World, Standard and Flying Cameras (at user-defined nodes).

- **To dolly a Camera up, down, left or right using the mouse**, hold down CTRL and click-and-drag with the left mouse button in the Shaded view or the shaded pane of Quad view.
- **To dolly a Camera in or out using the mouse**, hold down CTRL and spin the mouse wheel up or down while the Shaded view or the shaded pane of Quad view is active.
- **To dolly a Camera using the keyboard**, hold down CTRL and tap the arrow keys while the Shaded view or the Shaded pane of Quad view is active.
- **To dolly a Camera in or out using the keyboard**, hold down CTRL and tap the Page Up or Page Down keys while the Shaded view or the shaded pane of Quad view is active.

Important: Dollying a Flying Camera will physically alter the shape of its Camera Path.

Notes:

- This is the second of two ways in which a Flying Camera can be modified at a user-defined node—pivoting is the other.
- Flying Cameras cannot be dollied at nodes where Behaviour was set to *Follow Path* or *Follow Target*. In cases where something is "interfering with the shot" and the Flying Camera at such a node must be dollied, simply change the Behaviour at that

node to user defined—the Camera's current position and orientation will not be altered by the Behaviour change.

Roll

Rolling the Camera causes it to tilt left or right so that its bottom edge is no longer "parallel to the ground." This control applies to all Cameras, at all times.

- **To roll a Camera using the mouse**, click and drag left or right with the middle mouse button (i.e., the mouse wheel) in the Shaded view or in the shaded pane of Quad view.
- **To roll a Camera using the keyboard**, hold down ALT and tap the Page Up or Page Down keys while the Shaded view or the shaded pane of Quad view is active.

Zoom

Zooming the Camera causes its field of view to increase or decrease. This control applies to all Cameras, at all times.

- **To zoom a Camera using the mouse**, spin the mouse wheel up or down while the Shaded view or the shaded pane of Quad view is active.
- **To zoom a Camera using the keyboard**, tap the Page Up or Page Down keys while the Shaded view or the shaded pane of Quad view is active.

Creating a camera from shaded view

If you have set up a "perfect shot" in the shaded view, you can quickly and easily save the view to a camera. For more information on cameras, see "Drawing cameras" on page 164.

To create a camera from shaded view

1. In the shaded view, use your mouse to arrange the view that you want to see.
2. Once you are satisfied with the viewpoint, right-click in the shaded view and select **Save View as New Camera**.

   Tip: Alternately, on the Camera toolbar, you can click the *Save View as New Camera* icon. 
3. In the window that appears, type a name for the new camera, and then click *OK*.

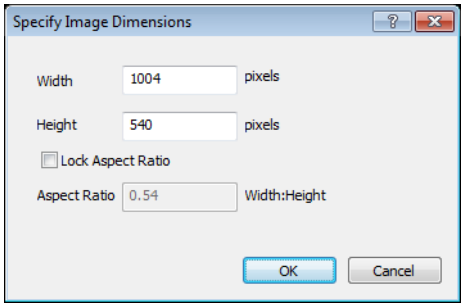
   Result: The camera is now available from the *Current Camera* drop-down box on the Camera toolbar.

To create an image from shaded view

If you have set up a camera or view of the shaded view that you would like to save as an image, you can do so by using the Export to Image feature.

1 Right-click in the shaded view and select **Export to Image**.

Result: The Specify Image Dimensions window appears.



2 Use the appropriate boxes to adjust the height and width of the image.

Note: The height and width correspond to the current dimensions of the shaded view.

3 To lock the aspect ratio of the dimensions, click to enable the *Lock Aspect Ratio* checkbox. If you want to change the aspect ratio, enter the new figure in the Aspect Ratio box.

4 Click *OK*.

5 Select a file name, location, and file type, and then click *Save*.

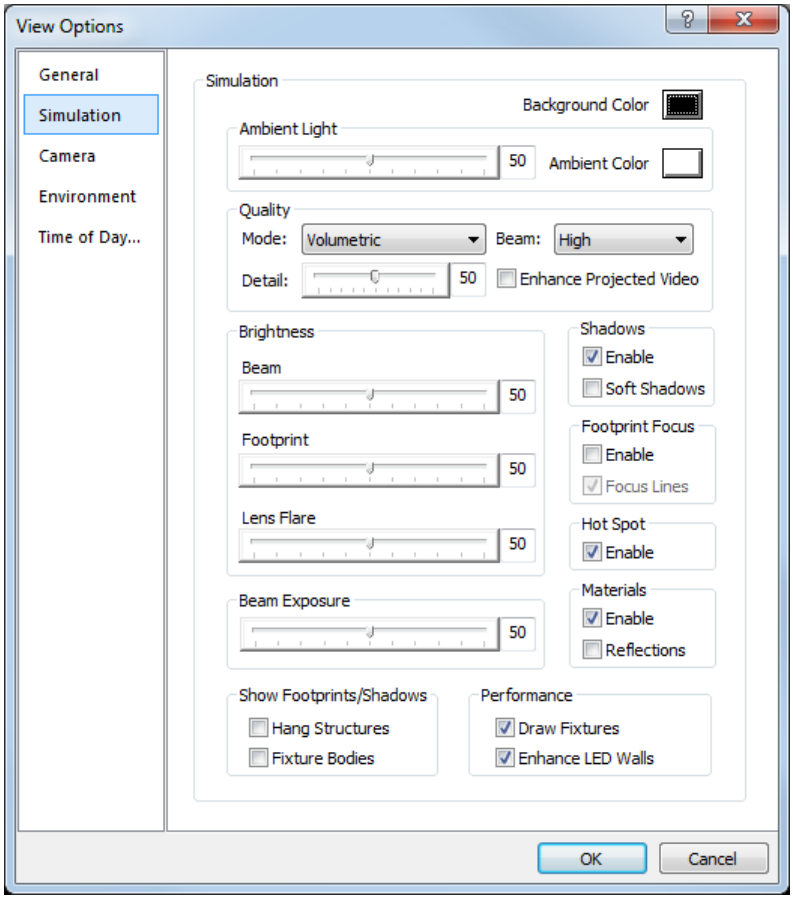
Modifying shaded views

The properties of a shaded view affect the view point, fixture beam quality, and ambient light. These settings are modified in the *View Options*.

To modify a shaded view

1 In shaded view, from the **Options** menu, choose **View Options**.

Result: The *View Options* window is displayed.



Tip: You can also use the *View Options* tool  on the *Standard* toolbar to open the view options.

2 Modify options as desired.

3 Click *OK*.

General tab

Options on the *General* tab affect the scene that is displayed in the Shaded view.

- *Follow User Scene*: Select this check box to use the scene that is displayed in the drop-down list on the Scene toolbar. Click to clear the check box, and then select the desired scene from the *Scene* drop-down list. To learn more about scenes, refer to “Scenes” on page 223.
  Tip: You can also change the current scene at any time by using the *Scene* toolbar.
- *View Styles*: These options allow you to define how the Shaded view appears.
  Note: These options do not impact the Shaded view performance.
  - *Object Bounding Boxes*: Select this option to show green bounding boxes around all objects in the scene. This view style was designed to be used in conjunction with the Consolidated Mesh feature (for details see “Consolidated Mesh” on page 252) and enables you to visually analyze an object’s complexity.
  - *Disabled Textures*: Select this option to hide (not remove) all textures used in a scene. This option is useful when verifying why a particular texture doesn’t seem to sit properly on an object and you want to check the underlying geometry for anomalies.

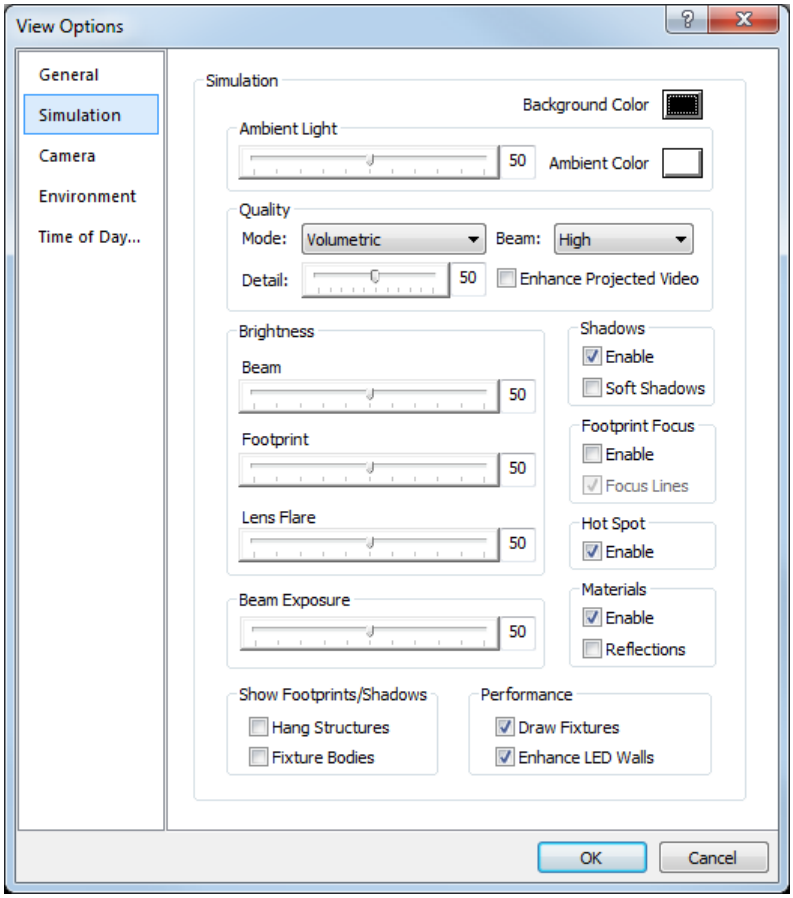
- *Wireframe Geometry*: Select this option to display all objects in a scene (except beams) as wireframe outlines instead of solids. This feature helps you analyze the complexity of a scene so you can alter objects as necessary to speed up rendering times and minimize file size. For example, when you see complex objects in your file (those shown with a high-density wireframe), you can optimize them by reducing their polygon counts, either with the Consolidated Mesh feature or by re-importing a less complex version.

- *Simulation Information*: These options allow you to define how simulation information is presented.
  - *View Statistics*: Enable this option to see information about the scene in the upper-left corner of the Shaded view. Note that enabling this view style will not modify the look of the scene. The information displayed is based on the Layers and/or Scene that are enabled. The following statistics are displayed:
  - *FPS*: Shows the framerate (frames per second) as the Shaded view is altered by moving the image with your mouse (which affects the fixtures in your file), changing Looks in Design mode, or by incoming DMX in Live mode. Note that when the Shaded view is static (as it is most of the time), the FPS field displays “n/a”.
  - *Number of Objects*: Provides information about the total number of visible objects.
  - *Number of Beams*: Displays the number of beams currently appearing in the Shaded view. Note that it is normal for the number of beams to be different from the number of fixtures. For example, a three-cell Cyc fixture will count as a single fixture, but when all three cells are on, the number of beams will read “3”.
  - *View Complexity*: Displays information about the complexity of the Shaded view. The four complexity levels—low, medium, high, and very high—are meant to help you understand how complex the current scene is. When the complexity level is high or very high, it is strongly recommended that you consider optimizing the file. For example, you could use the Consolidate Mesh feature to reduce the number of polygons in a highly-detailed model. For details, see “Consolidated Mesh” on page 252.
- *Camera Information*: Click to view information in Shaded about the Cameras that you have drawn.
- *Performance Graph*: Click to view a performance graph in Shaded view.

- *Visual References*: These options allow you to define how visual references are presented.
  - *Wireframe Selection Bounding Box*: Enable this option to highlight fixtures selected in wireframe view when the view is switched to shaded view.
  - *World Grid*: Click to show or hide the World Grid in Shaded views.
  - *World Axis*: Click to show or hide the World Axis in Shaded views.
  - *Gyroscope*: Click to show or hide the Gyroscope in Shaded views. The Gyroscope allows for continuous orientation monitoring while moving around in Shaded view.

Simulation tab

Options on the *Simulation* tab affect the view point, beam quality, and ambient light levels.



- *Ambient Light:* Use the slider to increase or decrease the amount of ambient light level shown in the shaded view. The percent box indicates the degree of ambient light you have chosen.
  - *Ambient Light*: Ambient Color modifies the color of the ambient light. For example, use this feature if you'd like the Ambient light in your file tinted yellow or blue to match the type of house lighting in your venue. By default, the Ambient Color is set to white.
  - *Background Color:* Modifies the background color used in the shaded view. This color is visible when you zoom far enough away that you can see beyond the venue.

Quality controls the degree to which you want to emphasize beam quality over performance.

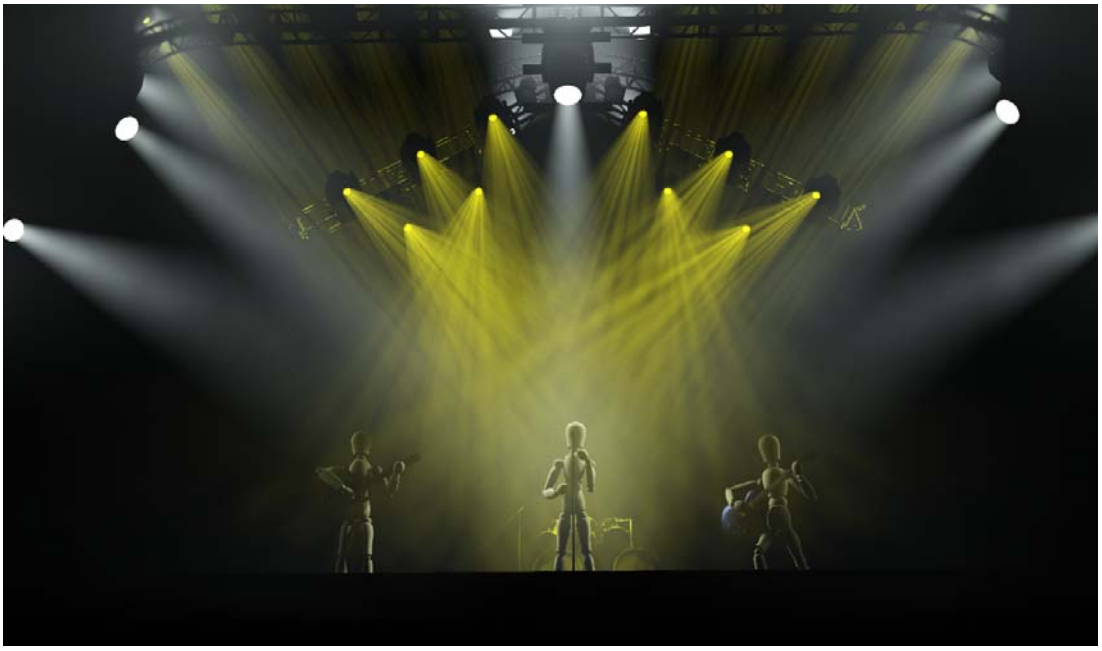
- *Mode*: Choose one of the following beam simulation modes. Note that when you select this value, and then choose the beam control options, you will automatically see the results in the Shaded view, enabling you to adjust them as necessary before you click *OK* to save your changes:

  Enhanced: Enhanced is the default beam mode. Select to enable enhanced beam control options affecting beam quality.

  Volumetric: This mode delivers higher quality visualization of beams then enhanced beams. Improved lens flair effects and beam throws are just some of the noticeable improvements. Due to the required system resources need to run Volumetric beams, system performance may be effected depending on system running WYSIWYG.



An example of a scene using Enhanced Beams in Shaded View.



An example of a scene using Volumetric Beams in Shaded View.

Note: Deferred Rendering is required to run Volumetric Beams. If this option is not available, it means the graphics card on your computer does not support Volumetric beams.

- *Detail*: Use the slider to increase or decrease the level of details displayed in *Volumetric* mode. An increase in details will result in a decrease in performance. The default detail setting is 0.

  Note: For most users, an increase in detail will not be noticeable, except in certain camera positions and situations.

- *Beam:* Lower beam quality affects the beams' footprint, sometimes making them appear blurry. While it is recommended that this value be set to *High*, if you notice a reduction in performance, particularly when viewing a complex scene with many moving lights, you can increase performance by lowering the quality to *Medium* or *Low*. Note that if you find it necessary to use Low quality for acceptable performance, it is recommended that you upgrade to a more powerful system.
  - *Enhance Projected Video*: This option is available only when High Beam Quality is selected. It should only be used when video is being projected, not when video is being used on a Screen, Surface, Object, or LED Wall. Enabling this feature decreases Shaded View performance, but increases the visual quality of the footprint (video) created by the projector.
- *Brightness*: Use the Brightness sliders (or type a value in the corresponding box) to control the visual brightness of beams, footprints, and lens flares. For example, a scene with many lights may be better visualized when some of the beams' components are less bright. Note that brightness settings affect all beams in the scene equally and are for aesthetic purposes only; the sliders are not connected to the fixtures' intensity.

  The Brightness sliders enable you to balance the brightness values for different scenes, each scene varying based on the density and intensity of its fixtures. These values are saved into the scene file and will, therefore, change when new scenes are loaded.

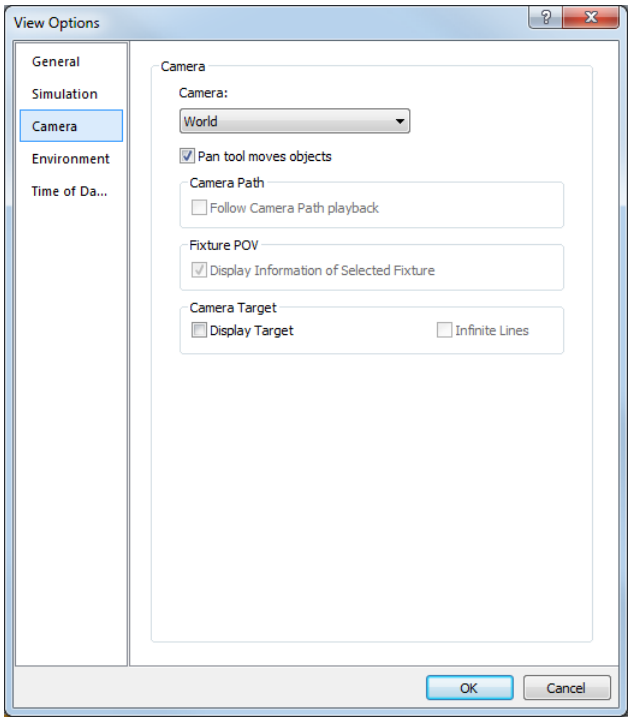
  Note: If the scene does not show any beams or footprints, ensure that none of the brightness sliders is set to zero.
  - *Beam*: Controls the brightness of the beam cone. A scene with many overlapping beam cones will tend to saturate, which can be reduced by decreasing the brightness.
  - *Footprint*: Controls the brightness of the beam footprint. A scene with many overlapping footprints will tend to saturated, which can be reduced by decreasing the brightness.
  - *Lens Flare*: Controls the brightness of the lens flare, which is the dominant visual effect when the camera is aimed directly into a fixture's lens. In addition, when you are behind a fixture that is directed away from the camera, the lens flare component is used to create a simulated beam cone.
- *Shadows*
  - *Enable*: Click *Enable* to display shadows. Turning on Shadows creates a more realistic Shaded View, but adds extra complexity, which may also slow down performance as more calculations are required.
  - *Soft Shadows*: Click *Soft Shadows* to display higher quality soft shadows. Use this option to soften the shadows for best visual quality, for example when capturing a screenshot of a look.

  Note: Enabling *Soft* shadows while running cues in Live mode is not recommended, as it will drastically slow down performance.
- *Footprint Focus*: Click *Enable* to toggle the footprint focus functionality. This option must be enabled to visualize the footprint focus, otherwise, fixtures for which you have defined footprint focus will not display. This toggle also offers an easy way to turn off the display of the footprint focus when not needed (for a performance boost) without losing the footprint focus settings of the fixtures. Click *Focus Lines* to toggle the display of focus lines, when the Footprint Focus option is enabled for conventional fixtures in CAD mode (fixture's Properties) and/or automated fixtures in Design mode (Footprint Designer tool).
- *Hot Spot*: Click *Enable* to toggle hot spot functionality. Hot spots are usually noticeable with conventional fixtures, which have lenses that refract the beam non-uniformly; the center of the beam is bright and it gradually drops off moving outward to the edge of the beam. Conversely, the majority of automated fixtures have a different type of lens that corrects the beam refraction to have a more uniform footprint.

- *Materials - Enable*: Click *Enable* to toggle materials functionality. When a material is applied to an object, it affect how the object interacts with light. This interaction is visible in shaded view and in rendered images from the WYSIWYG Render Wizard.
- *Materials - Reflections*: Click *Enable* to toggle reflections on materials. When a reflection is applied to an object, it will reflect light depending on the material settings. This interaction is visible in shaded view and in rendered images from the WYSIWYG Render Wizard.
- *Beam Exposure*: Use this slider to adjust the overall exposure of the beams. The default value is 50 which is the baseline value generated by the values from the library definitions. Increasing the exposure causes all beams in the scene to become brighter exponentially. This may be required if you feel all beams seem to dim. Conversely, decreasing the exposure causes all beams in the scene to become dimmer exponentially. This may be required if you feel all beams seem to bright/ saturated. If you wish to correct the relative brightness of one fixture, it is best to adjust its beam brightness by going into Fixture's properties and using the Beam Intensity Multiplier on the Beam options tab.
- *Show Footprints/Shadows*: You have the option of showing footprints and shadows in the Shaded view for one or more of the following two object types:
  - *Hang Structures*: This option controls whether hang structures like pipes and truss will cast shadows and receive footprints in the Shaded view. It is extremely useful for checking effects or looks such as truss warmers, but it is recommended to keep this option disabled in order to increase performance.
  - *Fixture Bodies*: This option controls whether the bodies of fixtures will cast shadows and receive footprints. This feature is useful when checking to see if certain fixtures are blocking the beams of other fixtures, but it is recommended to keep this option disabled in order to increase performance.
- *Performance*: You have the option of improving system performance by adjusting the following two object types:
  - *Draw Fixtures:* When selected, the fixture body will be drawn in shaded view. When deselected, the fixture body will not be drawn in shaded view, only the beam, which will reduce the amount of geometry being drawn in shaded views and offer slight performance improvements.
  - *Enhance LED Walls*: Click to improve the visual appearance of LED Walls. If there are many large LED Walls in the document, then performance may decrease as a result of having enabled this option; turn this option off and check whether performance improves.

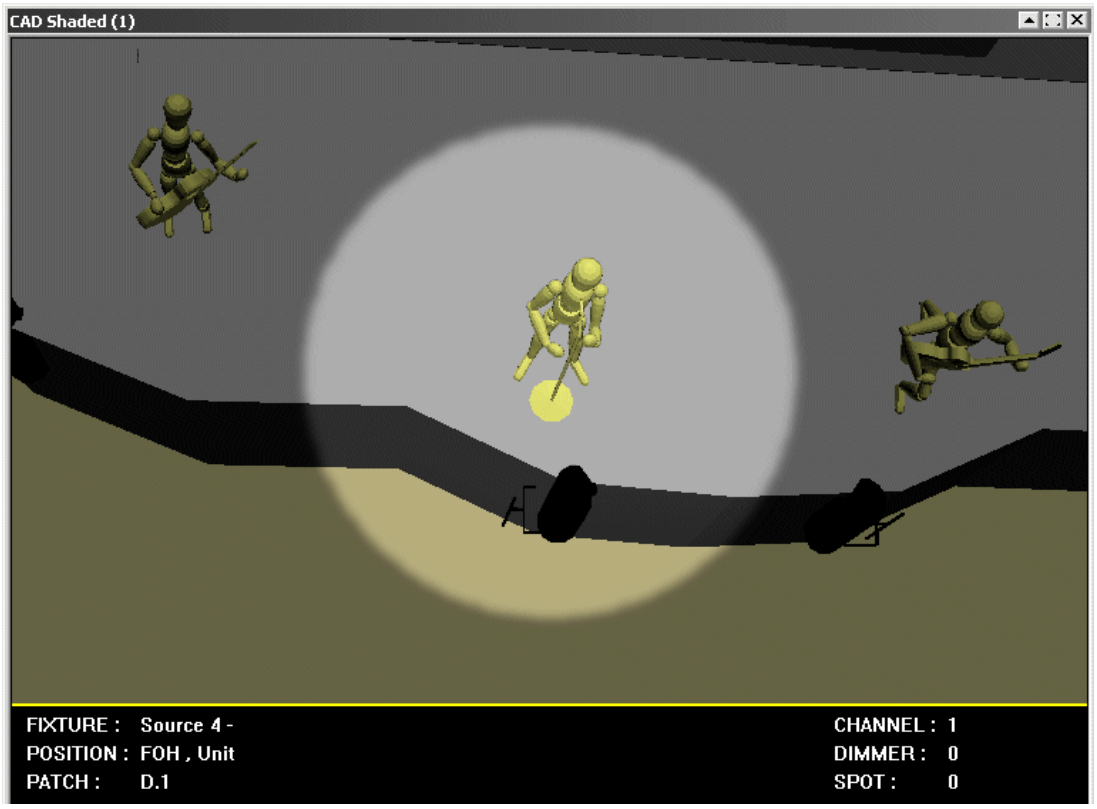
Camera tab

Options on the Camera tab affect the Cameras in your file.



- *Camera*: Select a camera to change the point of view. Click *Reset to Defaults* to remove any previous association with a camera and to display the original view point.
  Note: You can also control the camera view point with the camera toolbar. For details, see “Using the Camera toolbar” on page 165.

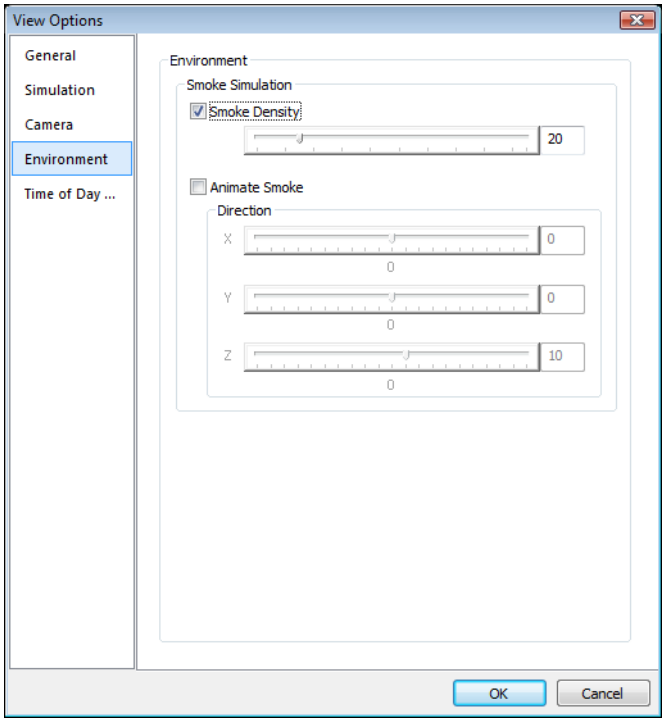
  Pan tool moves objects: Select this check box if you want to rotate the model on the target. Click to clear the check box if you want the pan tool to rotate the camera around the target. You can use the right, left, up, and down arrow keys in addition to the mouse button to pan to the desired location.
- *Fast Background Rendering*: Select this check box if you want to render the beams and not the background. The background is displayed as a pasted image and the beams are rendered on top of the background.
- *Display Information of Selected Fixture*: Select this check box to display information about the selected fixture at the bottom of the view.
  Note: This option becomes active when the Camera drop-down is set to *POV of selected fixture*, and only works if one fixture is selected. (When multiple fixtures are selected, the camera reverts to its previous camera setting and no fixture information displays).



- *Display Target*: Select the check box to show the focus point of the selected camera in the shaded view. Choose the color of the displayed target by clicking the *Color* box, and then selecting the color from the resulting palette. The target is visible in shaded views only.
- *Infinite Lines*: Select this check box to show the camera target lines of the selected camera in the Shaded view, extending the camera target lines to infinity

Environment tab

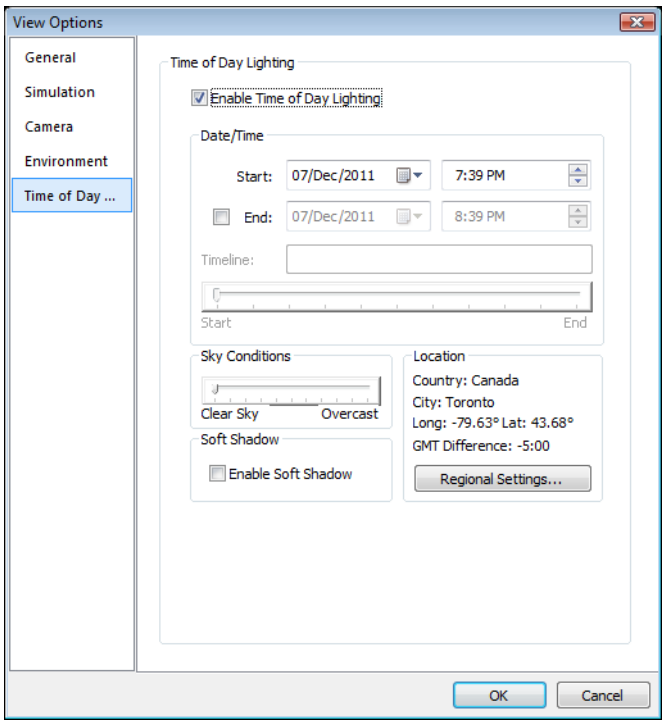
Options on the *Environmental* tab control non-uniform atmospheric effects in beams.



- *Smoke*: Beams can display non-uniform atmospheric effects in their beam cones.
  - *Smoke Density*: Click *Smoke Density* to display a non-uniform smoke suspension in the beam cone. Use the slider to change the density of the smoke.
  - *Animated Smoke*: Click *Animated Smoke* to make the non-uniform smoke move, and then specify the direction and speed using the X, Y, Z sliders. When the slider is in the middle at 0, smoke will not be animated in that axis. Move the slider off 0 to animate the smoke in that direction. The further away the slider is from 0, the faster the smoke will travel. (For example, to make the suspended smoke move in an upward direction to simulate a rising smoke effect, move the Z slider to the positive (right) side of 0, leaving the X and Y sliders at 0. To increase the speed at which the smoke is rising, move the slider further along on the positive scale.)

Time of Day Lighting tab

Just as you can simulate time of day/outdoor lighting when setting up your renderings in the Render Wizard, you can now achieve the same effect in the Shaded view in real time for all events that take place outside. Just like when you are setting up a rendering, in order to achieve the correct outdoor lighting conditions in the Shaded view, you must define the geographical location for your event and set the direction North. For details, see the “Regional settings tab” on page 401 and “To draw a compass” on page 170 respectively.



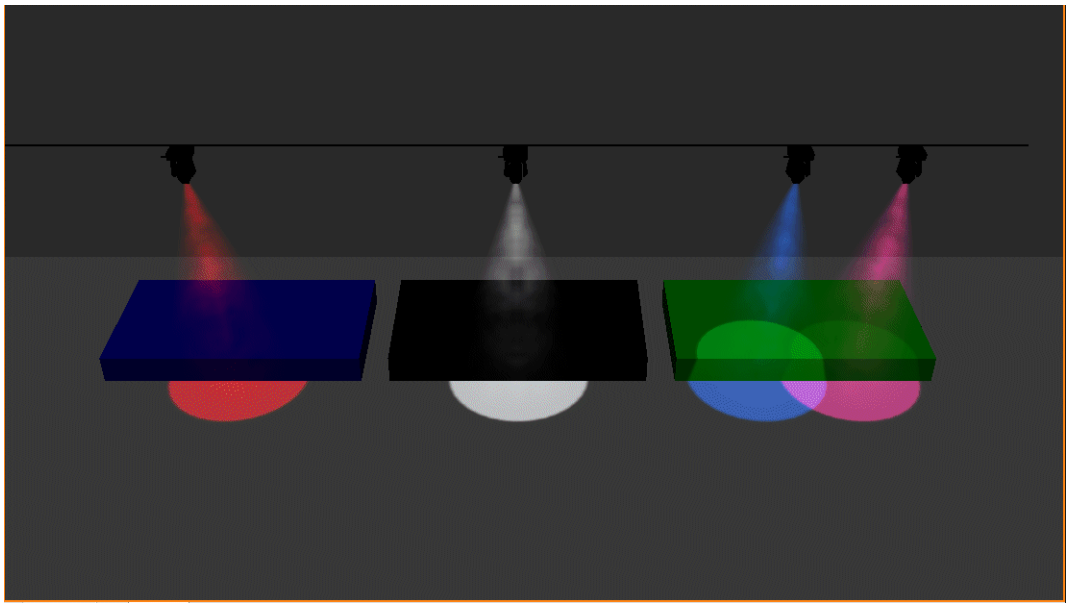
- *Enable Time of Day Lighting*: Select this check box to consider outdoor lighting conditions in the Shaded view, and then specify the lighting conditions below. The Shaded view adjusts accordingly, in real time.
  - *Date*: The date determines the position of the sun and/or moon and the resulting amount of light available. Type the date when the event will take place.
  - *Time*: The time determines the position of the sun and/or moon and the resulting amount of light available. Type the time when the event will take place.
  - *End*: Click this box if you want to specify an end time for the event, and then type the date and time in the boxes provided.
  - *Timeline*: By defining Start and End Dates/Times, you can check lighting conditions between those times. Use the Timeline slider to adjust the current Shaded view time.

- *Sky Conditions*: The amount of light available determines the brightness of the resulting image in the Shaded view. Use the *Sky Conditions* slider to adjust the quality of light (cloudy or clear).

- *Enable soft shadow*: Click this check box, and then use the slider to set the softness of shadows that are created by sunlight. The softer the setting, the less sharp the resulting shadow.
- *Regional Settings*: Click to set the geographic location of your show. You can either select the country and city or enter the latitude and longitude of the location. These settings are used in Shaded views and renderings that use environmental options

Beam quality color theory

The footprint technology of the beam simulation in our Shaded Views calculates footprints using color modulation, which is based on color theory. When a light beam strikes an object, some of the light is absorbed and some is reflected. The reflected color that the human eye can see is a subset of the source light spectrum that was not absorbed by the object.



For example, when a red light is shining on a blue riser there will not be any light reflected back to the human eye. The blue riser absorbs all colors that are not blue (its own color). Since there is no blue in the red source light, the light is completely absorbed. This means no footprint is displayed on the blue riser in our visualization.

Another scenario affected by this modulation would be a white beam hitting a black object. The white light contains all colors in the spectrum, while pure black is the total absence of color. As a result, the black object absorbs the entire spectrum of color and therefore reflects no light. This means no footprint is displayed on the black riser in our new visualization.

The next scenario shows two beams, a non-pure red and a non-pure blue, hitting a pure-green riser. The pure green riser absorbs all color components from the beam except for the green component and therefore reflects only green light. As we can see, the blue beam has a bit more green in its spectrum than the red beam, as it is slightly brighter than the red beam footprint. This example also shows that the portion of the footprints which fall on the grey floor result in the reflected color matching the beam

colors. Using color theory, grey which has an equal amount of red, green and blue would therefore absorb an equal amount of the beam color components as well as reflect an equal amount of these colors, resulting in the reflected color (footprint) being the same as the beam color.

Note: If you have a black stage, the footprints do not display on the stage because they are totally absorbed. To display footprint colors on black, use an unpure black or dark grays instead.

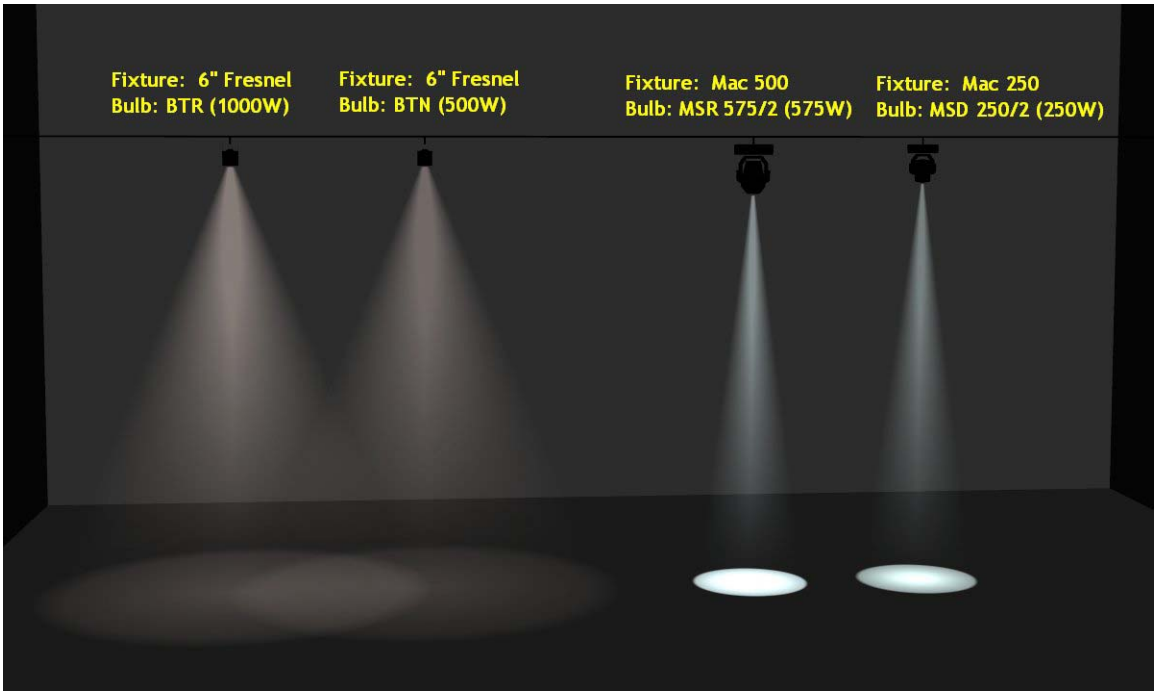
Color Temperature and Inverse Square Law

When in Shaded view, color temperature and inverse square law calculations are applied to all beams. These features create a greater degree of realism to Shaded view visualizations.

Color temperature calculations

For beams, the color temperature calculations take into account both the type of bulb used in the fixture and its wattage. For example, the beam of a fixture with an arc bulb displays with a blue tint, while the beam of a fixture with a tungsten bulb displays with a yellow tint. Likewise, the beam from a fixture using a 1000W bulb will be brighter than the same fixture using a 500W bulb, but, naturally, not twice as bright.

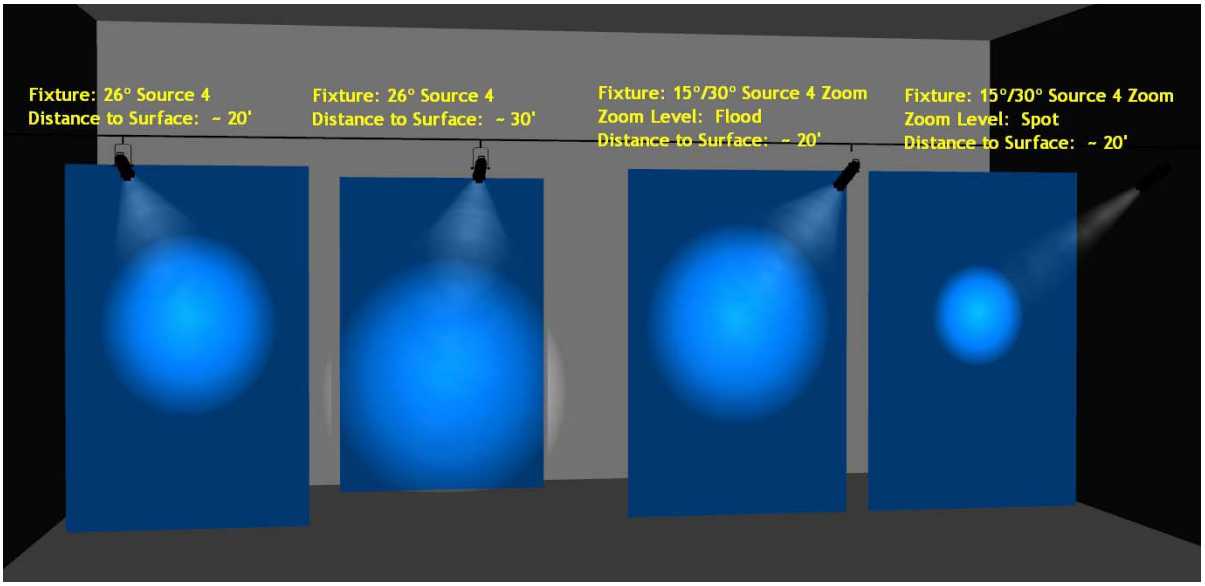
In the graphic below, note the difference in appearance of each of the beams: as the wattage increases, the beam appears brighter, and as the bulb differs, so does the tint of the beam.



Inverse Square Law Calculations

The inverse square law calculations apply to beam footprints, providing a realistic intensity falloff that is proportional to the beam angle and distance from source to target. For example, the footprint of a fixture lighting a flat surface 20’ away appears much brighter than if the same fixture were lighting the same flat surface at 30’. Similarly, for zoom fixtures, the intensity of the footprint is automatically adjusted when the beam angle is changed (by zooming in or out).

In the graphic below, note the difference in appearance of each of the footprints: as the beam gets narrower, the footprint appears brighter.



Chapter 5

CAD mode



CAD mode is where you create your show drawings. This includes drawing your venue, set pieces, lighting positions, focus positions, and lighting fixtures. CAD mode operates like many other CAD programs, so many of the concepts will be familiar to those who have used a computer-aided drafting program before. WYSIWYG adds features that are specific to the entertainment industry, such as a comprehensive 3D library containing truss, lighting equipment, lighting accessories, props, musical instruments, and various human figures.

In this chapter

Layout tabs	97
The CAD environment	98
The Library Browser	133
Drawing objects	138
Editing objects	185
CAD tools	226
Hang structures	257
Hanging and focusing fixtures	274
Color, gobos, and accessories	315
LED Wizard	322
Video Manager	335
Image Manager	343
The Flight Case	351
Rendering	354



Layout tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The CAD mode contains the following layouts:

- Wireframe - The work area displays a full-screen wireframe view.
- Quad - The work area is divided into quadrants, three of which can be modified to show plan, front, side, or isometric wireframe views. The lower-right quadrant contains a shaded view.
- Flight Case - The Flight Case is displayed in a section of the work area along with a wireframe view.
- Shaded - The work area displays a full-screen shaded view.

The CAD environment

Scales

When drawing in WYSIWYG, you are drawing in real scale (1:1). When you create a drawing in CAD, you are generating a virtual representation of your real set-up. If you were to do this on paper by hand, you would need to draw a scaled-down version of your space. However, since there are no paper size limitations in CAD mode, you can draw your venue, sets, pipes, trusses, and lighting fixtures in real scale.

Scaling down for printing purposes is done during print set-up and in the Presentation mode when creating plots. These settings allow you to print your drawings in whatever scales are necessary without having to redraw anything.

Coordinate system and origin

When working in CAD you are working in a 3D environment (even in WYSIWYG Report). Objects are drawn as 3D objects, with width, depth, and height values using the Cartesian coordinate system of 3 working axes X, Y, and Z. The point where the 3 axes meet is called the origin and the value of X, Y, and Z is 0 respectively (0,0,0).

By default, the origin is set at the center point of the WYSIWYG venue that you insert. Inserting venues is discussed in “Drawing a venue” on page 140. You can reset the origin to another point in your drawing; you can set a user origin so that a different point will assume the values (0,0,0).

To set the user origin

1 From the **Tools** menu, choose **Set User Origin**.

Tip: You can also use the *Origin* tool on the *Tools* toolbar. 

2 Click a point on your drawing or type in the coordinates (X,Y,Z) that will assume the values 0,0,0. This point will remain 0,0,0 until you change it again.

To reset the user origin

From the **Tools** menu, choose **Reset User Origin.**

Result: This resets the origin back to the WYSIWYG default.

Coordinate notation

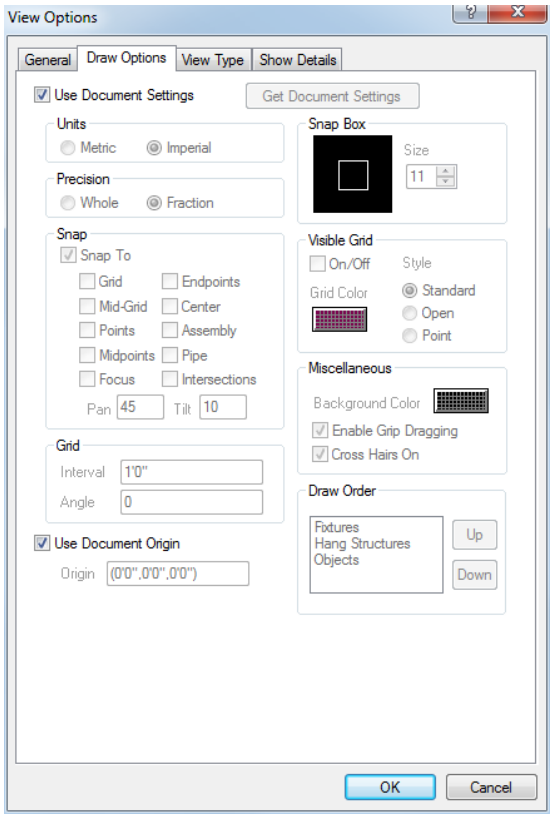
When entering coordinates in WYSIWYG, you can specify a location or distance in either inch fractions (to the sixteenth of an inch) or millimeters. For example, you can specify a measurement of 1’6”3/16”, which translates to 1 foot, 6 and 3/16 inches.

To set default options

1 From the **Options** menu, choose **View Options**.

2 Click the *Draw Options* tab.

3 Clear the *Use Document Settings* check box to modify the default setting for *Precision*.



4 Based on your choice for measurement units, you can choose to display the drawing resolution in whole numbers, fractions, centimeters, or millimeters.

- *Whole* - (Only visible if you choose Imperial units.) Indicates that the measurements are rounded and displayed to the nearest whole number.
- *Fraction* - Indicates that the measurements are displayed to the nearest sixteenths of an inch.
- *cm* - (Only visible if you choose Metric units.) Indicates that the measurements are displayed in centimeters.
- *mm* - Indicates that the measurements are displayed in millimeters.

5 Click *OK*.

Metric vs. Imperial

WYSIWYG enables you to use metric or imperial units at any time. You may choose to set a default type of unit, and you may choose to switch unit type on the fly.

You can also indicate whether you want these units measured in whole numbers, fractions (to the sixteenth of a inch), centimeters, or millimeters.

To set the default unit type

1 From the **Options** menu, choose **User Options**.

2 Click the *Draw Defaults* tab.

3 Click either *Metric* or *Imperial*.

4 Click *Whole, Fraction, mm* or *cm*.

Note: These choices vary based on whether you selected Metric or Imperial in step 3.

To switch unit types on the fly

Double-click the units display on the status bar.

Ruler Tool

The Ruler Tool helps you design your show file in the Wireframe views of the CAD Mode, providing a visual aid for coordinate reference and measurement. By default, the Ruler is aligned with the document origin and displays coordinate information along the top and left side of the view. Its scale matches the default grid scale.

As shown in the graphic below, the Ruler has different colors to represent different axes (X=Red, Y=Green, Z=Blue).

The Ruler dynamically updates as you zoom in and out. When you zoom in on your drawing, the precision of the ruler increases, displaying fractions and decimals; when you zoom out, the precision decreases to the point where both the grid and ruler disappear. The ruler matches the unit type currently set in your file, either Metric or Imperial. When working with larger venues, it is recommended that you increase the grid spacing so that the ruler will be visible when zoomed out further. The ruler matches the unit type.

Ruler icons

On the Quad tab (as shown above), each of the three drawing quadrants can have a different drawing origin, or View Origin. When you define a new view origin, you can move the zero position of the ruler (-0 0+) to correspond to this point.

Based on the origin in the active quadrant and the corresponding position of the Ruler, the icon in the upper left corner of the Ruler changes.

Icon	Description
	The Ruler's zero position (origin) is aligned with the Document Origin, which is set by default to be the center point of the WYSIWYG venue defined for the event.
	By default, all views are set to the Document Origin. However, you can set the origin for each view, thereby defining a View Origin. This icon appears when you have defined a View Origin for the current view and have aligned the Ruler's origin to it.
	This icon appears when you have set a new User Origin for the file by clicking **Tools** > **Set User Origin**.
	This icon appears when you have clicked **Move Ruler** (Set View Origin) from the **Ruler** menu and are in the process of defining your new origin.

The following graphic shows a Quad tab in which the Ruler origin is different in each view:

To turn the Ruler on and off

By default, the Ruler is turned on when you open a new WYSIWYG file. You can turn the ruler on and off while working in your file by using the Ruler Toggle icon on the CAD Options toolbar.

1 To show the CAD Options toolbar, right-click in the toolbars area and click to place a check mark beside **CAD Options**.

2 On the CAD Options toolbar, click the Ruler Toggle icon. 

Result: The Ruler appears or disappears.

Note: You can also toggle the ruler on or off by using the Draw Defaults tab in User Options. Click **Options** > **User Options > Draw Defaults**.

- To turn the Ruler off, deselect the *Rulers On/Off* check mark.
- To turn the Ruler on, ensure that there is a check mark in the *Rulers On/Off* check box.

  Click *OK*.

To change the Ruler's origin

By default, the ruler is aligned with the document origin, which means that the zero mark (-0 0+) starts at the Document Origin. However, you can move the Ruler from this default position to the origin of the current view. Note that doing so does not affect the origin or ruler of any other view.

1 In the Quad tab, click the quadrant in which you want to move the Ruler.

2 Click the icon in the upper left corner of the Ruler (or right-click on the actual Ruler) and select one of the following options:

- **Display Ruler at Document Origin**: Select this option to realign the Ruler's zero point with the Document Origin, which is set by default to be the origin of the WYSIWYG file. When the ruler is displayed at the Document/User origin, the background color of the ruler is white.
- **Display Ruler at View Origin**: Select this option if you want to realign the Ruler to the View Origin of the current view, if different than the original Document Origin. To indicate this change, the background of the Ruler changes from white to match the axis colors.
- **Move Ruler (Set View Origin)**: Select this option if you want to choose a new origin for the ruler. Then click the cursor on the point in your drawing that you want to designate as the zero mark for the Ruler. Once the Ruler is moved, you can use the previous two menu items (**Display Ruler at Document Origin** and **Display Ruler at View Origin**) to switch the ruler's position from the default position to the user defined position.

Note: By moving the ruler, you are indirectly setting a new origin for the currently selected view. In the past, this was possible by changing the Origin in a CAD view's View Options, or by clicking Tools > Set User Origin.

The missing coordinate

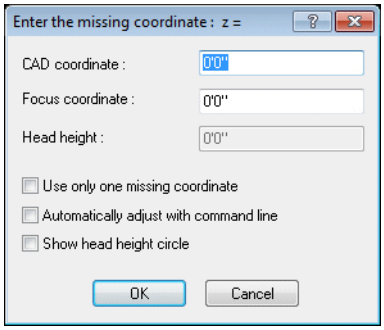
The missing coordinate (X,Y,or Z) is the coordinate whose value cannot be entered by clicking on the screen. The easy way to determine which coordinate is the missing coordinate in a wireframe view is to move the mouse around. Look at the status bar at the bottom of the working area. You will see only two values changing. The value that is not changing is the missing coordinate for that view or workplane. The missing coordinate is dependent on the plot type and the workplane selected. In the following example, Y is the missing coordinate as its value is set at 0'0".



Once entered, this value affects all subsequent objects inserted in the current view. For example, if the missing coordinate is set to 5' in a plan view, all objects are placed 5 feet off the floor (X,Y,5) until the missing coordinate is changed again. Please note that the missing coordinate only affects subsequent inserts.

To enter a value for the missing coordinate

1 Press TAB on your keyboard or click the *Missing Coordinate* tool on the *CAD Options* toolbar. 



2 In the *CAD coordinate* box, type the value for the CAD coordinate.

3 The *Focus Coordinate* tells WYSIWYG where to place the cursor (i.e., at which missing coordinate) when manually focusing fixtures. If you type any value other than 0 in this box, you will notice that the beam's centreline extends past the crosshairs when you drag the beam around with your mouse. This represents the distance between the Focus Coordinate and the floor (or whatever is stopping the beam).

4 If you want to use only the CAD coordinate for focusing and placing/editing objects, select the *Use only one missing coordinate* check box.

5 To automatically adjust the missing coordinate with any 3D coordinate value that you enter on the command line, select the *Automatically adjust with command line* check box.

6 Select the *Show head height* check box to display the coverage of beams at the head height selected in the *Head height* box.

7 In the *Head height* box, type a value to check the light coverage at a certain height (the height of the performer).

Note: This box is enabled when the *Show head height circle* check box is selected.

8 When you are finished with your selections, click *OK*.

Example

1 Change to the plan view in your drawing.

2 Press TAB, and then type a value of 0" for the missing CAD coordinate.

3 Insert a few objects on the stage. All of these objects are drawn resting on the stage.

4 Change the missing CAD coordinate to a height of 10’.

5 Insert a riser by clicking the Riser button on the toolbar.

6 Although we are in a plan view the riser has been drawn at the height of 10', as specified in the missing coordinate dialog box.

7 Change to a side view and you will see that the riser has been placed above the stage.

Tip:

- If you select the *Show head height circle* check box, and then type a value in the *Head height* box, all subsequently focused fixture’s beams will display both the coverage at the specified head height, as well as the footprint of the beam on the surface below.

Command line

The command line is an area in WYSIWYG where you can enter coordinates for the purpose of placing or editing objects in a document. The placement of objects can often be done quicker and with more precision using the command line.

Generally, coordinates are specified as X, Y, Z. You can, however, insert coordinates using either two or three values. When using two values, the third value will be assumed from the missing coordinate. For more information on the missing coordinate, refer to “The missing coordinate” on page 102.

The values that you specify in imperial mode are assumed to be in feet unless otherwise specified. Similarly, the values that you specify in metric mode are assumed to be in metres unless otherwise specified. At any time, you can specify values in both imperial and metric measurements (for example, 5”, 3 cm, 6’).

The following example illustrates the many different ways of using the command line.

Example

1 In a plan view, from the **Draw** menu, choose **Line**.

2 From the sub-menu, choose **Solid**, **Dot**, **Center**, **Hidden** or **Spline**.

3 Type 0,0 as the starting point of the line.

Result: When you start typing, the ComEdit toolbar is displayed.



4 Press ENTER to establish the first point of the line at the origin.

5 To set the next point at exactly X=5 and Y=5, type 5,5, and then press ENTER.

Result: A new line segment is drawn and it assumes Z from the missing coordinate.

6 To place the next point of the line 10 units to the right and 5 units up (in Y) from the last point, type @10,5, and then press ENTER.

Result: A new line segment is drawn.

7 To place the next point at exactly 5 feet, 3 metres, and 6 inches from the last point, type 5', 3m, 6", and then press ENTER.

Result: A new line segment is drawn.

8 When you are finished drawing the lines, right-click, and then select **Finish Line**.

Plot types

All objects occupy three-dimensional space. Different plot view types allow you to see and work with your drawing from different perspectives.

There are six types of plot views accessible from the CAD Options toolbar. The following definitions identify the working axes X, Y, and Z and the missing coordinate for each plot type. The missing coordinate can be defined as the axis for which a value cannot be set simply by clicking on the work space. For details, see "The missing coordinate" on page 102.



Plan View

Plan views display the plot from above looking down. This is similar to a plan view drawing on paper. In plan views the working axes are X and Y and the missing coordinate is Z.



Left View

Left views display the plot looking from the left side through the venue. This is similar to a section on paper. In left views the working axes are Y and Z and the missing coordinate is X.



Right View

Right views display the plot looking from the right side through the venue. This is similar to a section on paper. In right views the working axes are Y and Z and the missing coordinate is X.



Front View

Front views display the plot looking from the front side through the venue. This is similar to an elevation on paper. In front views the working axes are X and Z and the missing coordinate is Y.



Back View

Back views display the plot looking from the back side through the venue. This is similar to an elevation on paper. In back views the working axes are X and Z and the missing coordinate is Y.



Isometric View

An Isometric view is a 3D perspective drawing. Despite this, you are still limited to two working axes. In isometric views, the working axes and the missing coordinate are dependent on the workplane selected. The workplanes available are:

-  plan
-  side
-  front

If a plan workplane is selected, the working axes are X and Y and the missing coordinate is Z.

If a side workplane is selected, the working axes are Y and Z and the missing coordinate is X.

If a front workplane is selected, the working axes are X and Z and the missing coordinate is Y.

The crosshairs of your cursor change to reflect the selected workplane.

Note: Isometric views are not available in WYSIWYG Report.

Ortho

Ortho mode constrains movement and drawing to a direction parallel to the specified axis. When ortho mode is inactive, objects can be drawn or moved in any direction on the workplane. You can draw diagonal lines or move objects anywhere in the working area.



You can activate ortho at any time by clicking the ortho tools on the *CAD Options* toolbar or by right-clicking the **Ortho** label on the status bar. You can also press F8 on the keyboard or double-click the Ortho label on the status bar to enable Ortho or to toggle your last recorded ortho setting on or off.

Notes:

- The plot type determines the axes of movement: XY for plan views, YZ for side views, XZ for front and back views.
- To select the axes of movement with the toolbar buttons, click the appropriate two buttons to select both of them. For example, to choose the XY axis, click the *X* and *Y* buttons.
- To deselect any ortho buttons that you have chosen, press F8; to reset the same ortho selection as that which was previously made, press F8 again.

Snaps

Snaps are used to assist in the drawing and placement of objects. When a snap setting is active, the cursor will be drawn to the applicable snap point. You can have multiple snaps active at the same time. Snap selections can be made on the CAD Options toolbar or from the Tools menu.

All snap tools function as follows: If no command is currently active, you can adjust the running snap tool. This means that the snap type is always active. If you are in mid-command and you select a snap tool, it becomes a “one time only” snap type. After the next click, the snap returns to the running snap. At any time you can toggle all snaps on or off by pressing F9 or double-clicking on the SNAP text in the status bar.

-  Grid snap

Aligns the placement of objects to grid points. You can set the grid interval and angle in the *Draw Options* page of the *View Options* window.

-  Mid-grid snap

Aligns the placement of objects between grid points.

-  Point snap

Aligns the placement of objects to a point in the drawing.

-  Midpoint snap

Aligns the placement of objects to the mid-point of another line.

-  Endpoint snap

Aligns the placement of objects to the end-point of another line.

Fixture insertion points are considered points. It is therefore possible, using the Point snap, to align the placement of objects to fixtures (snap to fixtures). This is useful for dimensioning.

-  Intersection snap

Aligns the placement of objects to the intersection of lines, circles or arcs on the same plane.

-  Centerpoint snap

Aligns the placement of objects to the center-point of circles, arcs, or cylinders.

-  Assembly snap

Use assembly snap to group multiple truss objects together as you insert them. This ensures proper structural assembly. Refer to “Using Assembly snap with truss” on page 268 for more details about truss assembly.

-  Pipe snap

Constrains fixtures to placement on a pipe at a specific interval. For more information, refer to “Pipe snap” on page 279.

-  Focus snap

Orients one or more fixtures to a focus position. When you choose this value, you can also lock beam dragging to the incremental values of your choice, as specified in the *Pan* and *Tilt* boxes of the **Draw Options** tab (the defaults are increments of 45 degrees for pan and increments of 10 degrees for tilt). These are the values to which you want the fixture's beam to “snap” while manually dragging and focusing it. For example, if you set a value of 30 degrees, when you drag the fixture's beam, it will snap at 30 degrees, 60 degrees, 90 degrees, and so on. Note that any focus positions that you have set take precedence over the pan and tilt values (if you drag the beam over the focus position, it will automatically snap to the focus position instead).

Interactive mode

Interactive mode is an alternative method for drawing objects. Objects are typically drawn using a dialog box to set the object’s size (width, depth, and height, for example). The full-size object is then attached to the cursor so you can place it in the drawing.

An Interactive button enables you to switch modes. Interactive mode allows you to click an insertion point for the object first, and then drag to create the scope of the object as allowed by the view (XY for plan view, XZ for front view, and so on).

Once those dimensions are set, a dialog box may open to allow you to set the third dimension, if required. Interactive mode works with Risers, Cylinders, Circles, Arcs, Spheres, and Pipes. There are some special considerations when drawing pipes in interactive mode, as explained in “Drawing pipes” on page 162.

To set interactive as the default drawing mode

1. From the **Options** menu, choose **User Options**.
2. Click the *User* tab.
3. In the *Options* area of the window, click the *Interactive object creation* check box.

To set interactive mode on the fly

Press F11 on your keyboard or click the Interactive tool on the *CAD Options* toolbar.

Importing SketchUp files

SketchUp is a modelling tool that enables you to create realistic 3D models of many different types of objects and then save the files with an .skp extension. You can import these SketchUp files into WYSIWYG in much the same way as you import DWG/DXF files.

Before you import SketchUp files into WYSIWYG, note the following:

- While SketchUp supports two textures per surface, one each for the front and back, WYSIWYG supports only one texture for both sides of imported surfaces; when you are importing SketchUp files, WYSIWYG will automatically apply the front texture to both sides of the object.
- Since WYSIWYG does not support nested custom library items, when converting nested SketchUp components into custom library objects, you need only to import the top-most component.

Textures

When you import SketchUp files into WYSIWYG, the textures associated with these files are saved in a folder on your computer so you can apply them to imported surfaces. By saving them to a folder outside of WYSIWYG, you can use them repeatedly instead of only applying them to the surfaces that are within the current WYSIWYG document.

Upon import, you can choose the location for saving the associated texture files (or accept the default location). If there is already a texture with the same name saved in the location specified, WYSIWYG will prompt you to save the file with a new name.

Import options

When importing a SketchUp file into WYSIWYG, you have two options:

- You can *open* a file.
- You can *merge* a file into an existing WYSIWYG document.

If you *open* a SketchUp file while another show document is currently open, you are prompted to save changes to that document before another show document is opened. Only one show document may be open at a time. When you *merge* documents, you can add the contents of the SketchUp file to the contents of the current WYSIWYG document.

To open SketchUp files

This procedure creates a new WYSIWYG document out of the SketchUp file, prompting you to save changes to any current document that is open and close it before importing the file.

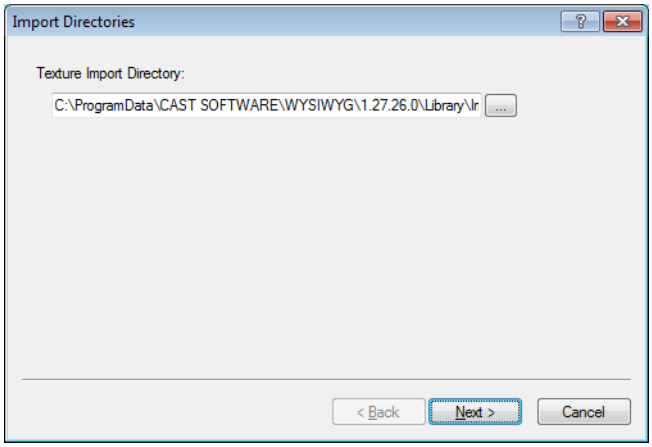
Note: Before you begin this procedure, you must have the SketchUp file saved on your computer.

1 From the **File** menu, choose **Open**.

2 From the *Files of type* box, select SketchUp files (.skp), and then navigate to the location where the file is saved on your computer.

3 In the browser, click the file name, and then click *Open*.

Result: The *Import Directories* window appears.



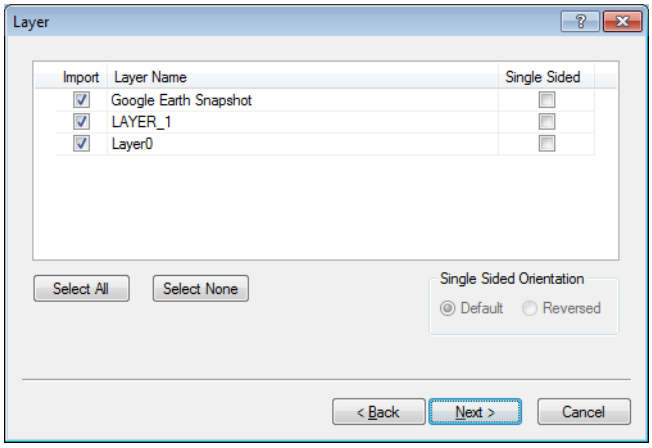
4 Click the browse button to specify the location where you want to save the textures that have been applied to the object in SketchUp, or accept the default location shown. By saving textures to a folder outside of WYSIWYG, you can use them repeatedly instead of only applying them to the surfaces that are within the current WYSIWYG document.

Note: If a default path is not shown, you must click the browse button and navigate to the desired folder. If there is already a texture with the same name

saved in the location specified, WYSIWYG will prompt you to save the file with a new name.

5 Click *Next*.

Result: The *Layer* window appears.



6 In the *Layer* window, select the layers that you want to import.

- If there is more than one layer listed, you can select multiple layers simultaneously by pressing *Ctrl* on your keyboard and then clicking on the desired layers.
- For each layer that you want to import, ensure that the *Import* check box is checked. If you do not want to import a certain layer, highlight it and deselect this checkbox. For example, if you have drawn both a venue and elements in SketchUp and you have put them on separate layers, you can choose to only import a particular element into the WYSIWYG file by selecting the appropriate layer in this window.

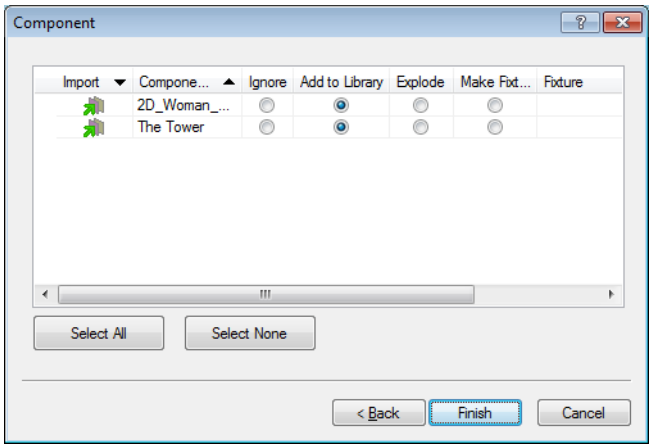
7 To import a single-sided layer, highlight it and select the *Single Sided* checkbox.

Note: Single-sided layers use less processor time to display in Shaded views and to render in the Render Wizard. However, they only appear properly in each of these views if the faces of the object are drawn facing outward. Unless you know how the SketchUp file was drawn, it is recommended that you leave *Single Sided* deselected by default when importing the layers so that the object appears correctly. Note that you will not be able to see “through” the walls of an imported double-sided venue when you rotate the image around in the Shaded view (unlike the WYSIWYG venues, which are single sided).

If the imported object does not appear correctly in the Shaded view, you can change its sidedness by using the *Properties* window. You can change it from single sided to double sided, or vice versa. You can also flip the object’s sides if the wrong side is currently facing outward. For details, see “To change an object’s sidedness” on page 203.

8 Click *Next*.

Result: The *Component* window appears.



9 Blocks in SketchUp are called components. If you have components in your drawing, you have the option to ignore them, explode them, convert them into a library item, or substitute WYSIWYG fixtures for the existing components.

a. *Import*: This icon indicates that the block is currently on a layer being imported. This icon indicates that the block is on a layer that will *not* be imported.

b. *Component Name*: The name of the block.

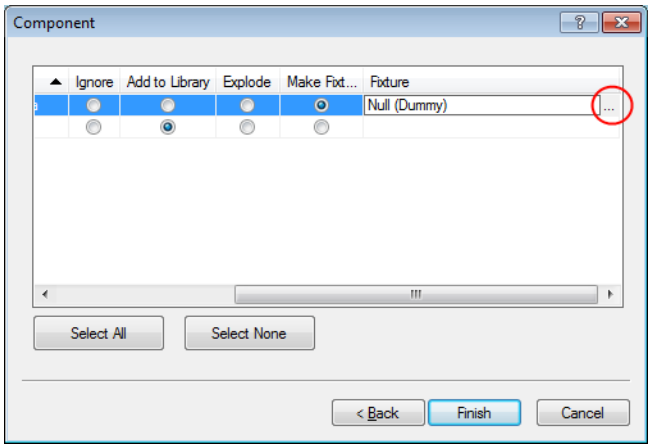
c. *Block Type*: This icon indicates that the block is a singular block and does not contain any sub-blocks. This icon indicates that the block is a nested block and contains one or many sub-blocks.

d. *Ignore*: Select this action to ignore the block. The block will not be added to the resulting file.

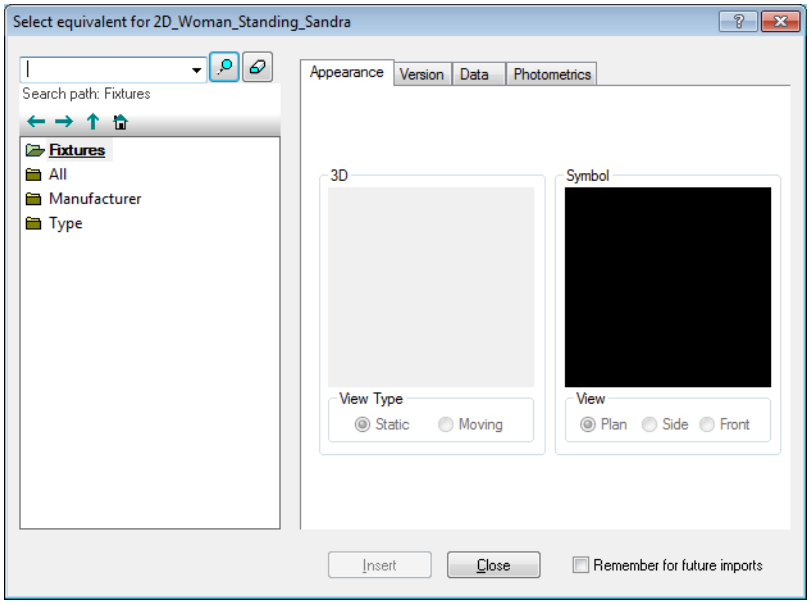
e. *Add to Library*: Converting a block into a library item imports the object and creates a duplicate of it to add to the library for future use. Note that the component will lose all surface properties, such as color and texture, and it will only be available in the library of the current document. To make it available globally, see “To create a custom library item” on page 137. When you convert SketchUp files into custom library items, they appear on the Library tab of the Library Browser within the SketchUp folder.

f. *Explode*: Exploding the component breaks it into its sub-components. That is, it breaks the block into its components. You must be careful of components that are made up of other components as WYSIWYG will only explode down one level.

g. *Make Fixture*: Converting a component into a fixture replaces the SketchUp component with a WYSIWYG fixture of your choice. Once you have selected this option, select the ellipsis in the neighboring column.



Result: The Fixture select window appears.

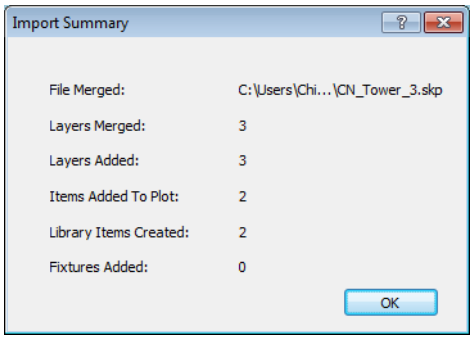


10 Select the fixture that you want to use to replace the block. If you find that you are always replacing a particular type of component with a specific fixture, you can set an option in WYSIWYG so that the substitution will automatically occur each time you import the component. For example, if you have files in which the 'PAR64M' block is always a PAR 64 fixture with an MFL bulb, you have the option of telling WYSIWYG to automatically import the component in that way. To change this option, edit the Import.lst file that is created in the Library folder (usually C:\Program Data\CAST Software\WYSIWYG\1.xx.xx.x\Library) once the first fixture is set to be remembered for future imports.

Note: WYSIWYG creates a pipe for every imported fixture since fixtures in WYSIWYG need to hang on a hang structure. If, however, fixtures are on a straight pipe represented by a straight line in the DWG file, WYSIWYG will convert the entire line into a pipe.

11 Click *Finish*.

Result: The Import Summary window appears.



12 Click *OK*.

To merge SketchUp files

This procedure merges the SketchUp file into the current WYSIWYG document, placing the object in the location you specify.

Note: Before you begin this procedure, you must have the SketchUp file saved on your computer.

1 Open the WYSIWYG file into which you want to add the SketchUp file.

2 From the **File** menu, choose **Merge**.

3 Click *Browse* to locate the SketchUp file on your computer.

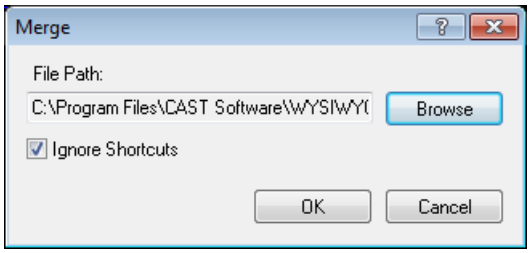
4 In the Merge window that appears, beside the File name box, select SketchUp Files from the drop-down list.

5 Browse to locate the file, select it, and then click *Open*.

6 Leave *Ignore Shortcuts* selected, and then click *OK*.

Note: Deselect this checkbox if you would like to keep the shortcuts (i.e., DESIGN mode Looks).

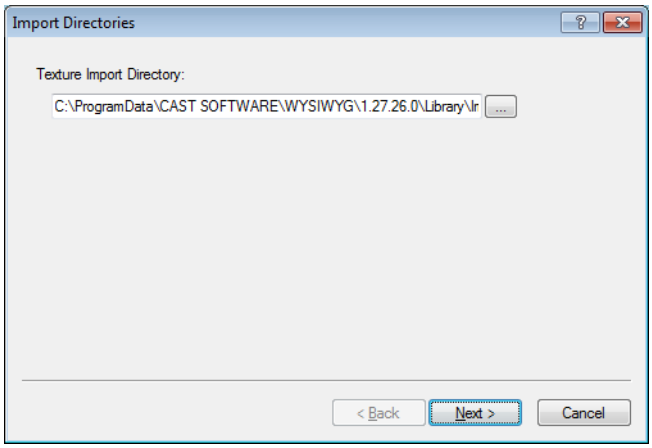
Result: The *Merge* window appears.



7 Click one of the options to choose the base point for the merged object, and then click *OK*.

8 In the browser, click the file name, and then click *Open*.

Result: The *Import Directories* window appears.

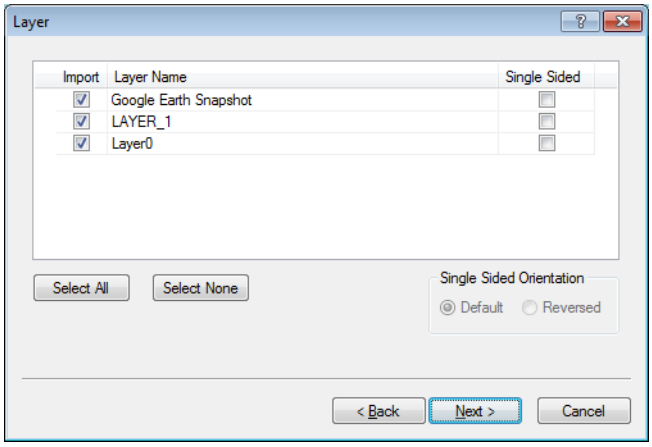


9 Click the browse button to specify the location where you want to save the textures that have been applied to the object in SketchUp, or accept the default location shown. By saving textures to a folder outside of WYSIWYG, you can use them repeatedly instead of only applying them to the surfaces that are within the current WYSIWYG document.

Note: If a default path is not shown, you must click the browse button and navigate to the desired folder. If there is already a texture with the same name saved in the location specified, WYSIWYG will prompt you to save the file with a new name.

10 Click *Next*.

Result: The *Layer* window appears.



11 In the *Layer* window, select the layers that you want to import.

- If there is more than one layer listed, you can select multiple layers simultaneously by pressing *Ctrl* on your keyboard and then clicking on the desired layers.
- For each layer that you want to import, ensure that the *Import* check box is checked. If you do not want to import a certain layer, highlight it and deselect this checkbox. For example, if you have drawn both a venue and elements in SketchUp and you have put them on separate layers, you can choose to only import a particular element into the WYSIWYG file by selecting the appropriate layer in this window.

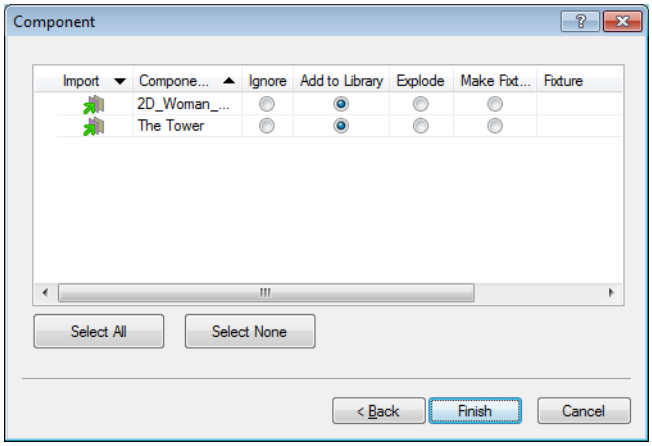
12 To import a single-sided layer, highlight it and select the *Single Sided* checkbox.

Note: Single-sided layers use less processor time to display in Shaded views and to render in the Render Wizard. However, they only appear properly in each of these views if the faces of the object are drawn facing outward. Unless you know how the SketchUp file was drawn, it is recommended that you leave *Single Sided* deselected by default when importing the layers so that the object appears correctly. Note that you will not be able to see "through" the walls of an imported double-sided venue when you rotate the image around in the Shaded view (unlike the WYSIWYG venues, which are single sided).

If the imported object does not appear correctly in the Shaded view, you can change its sidedness by using the *Properties* window. You can change it from single sided to double sided, or vice versa. You can also flip the object's sides if the wrong side is currently facing outward. For details, see "To change an object's sidedness" on page 203.

13 Click *Next*.

Result: The *Component* window appears.



14 Blocks in SketchUp are called components. If you have components in your drawing, you have the option to ignore them, explode them, convert them into a library item, or substitute WYSIWYG fixtures for the existing components.

a. *Import*: This icon indicates that the block is currently on a layer being imported. This icon indicates that the block is on a layer that will *not* be imported.

b. *Component Name*: The name of the block.

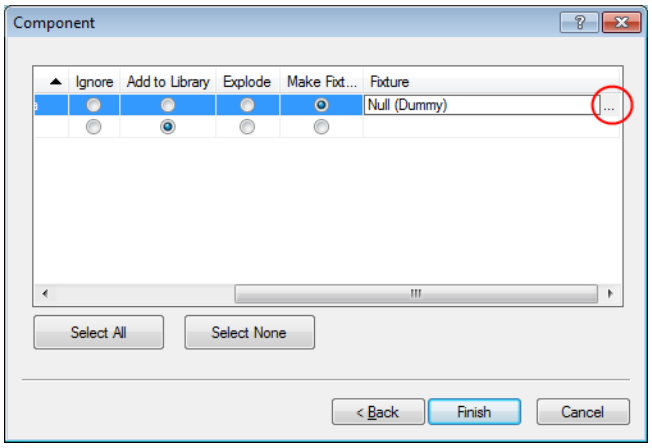
c. *Block Type*: This icon indicates that the block is a singular block and does not contain any sub-blocks. This icon indicates that the block is a nested block and contains one or many sub-blocks.

d. *Ignore*: Select this action to ignore the block. The block will not be added to the resulting file.

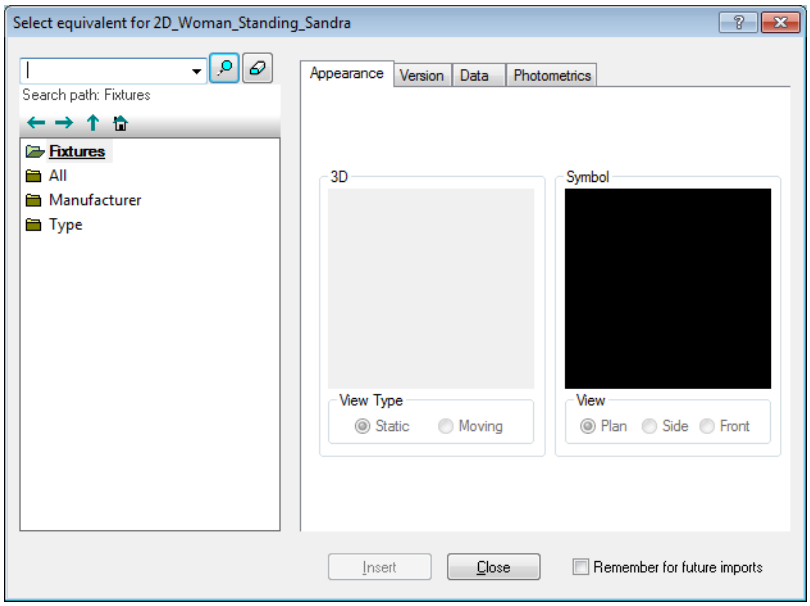
e. *Add to Library*: Converting a block into a library item imports the object and creates a duplicate of it to add to the library for future use. Note that the component will lose all surface properties, such as color and texture, and it will only be available in the library of the current document. To make it available globally, see "To create a custom library item" on page 137. When you convert SketchUp files into custom library items, they appear on the Library tab of the Library Browser within the SketchUp folder.

f. *Explode*: Exploding the component breaks it into its sub-components. That is, it breaks the block into its components. You must be careful of components that are made up of other components as WYSIWYG will only explode down one level.

g. *Make Fixture*: Converting a component into a fixture replaces the SketchUp component with a WYSIWYG fixture of your choice. Once you have selected this option, select the ellipsis in the neighboring column.



Result: The Fixture select window appears.

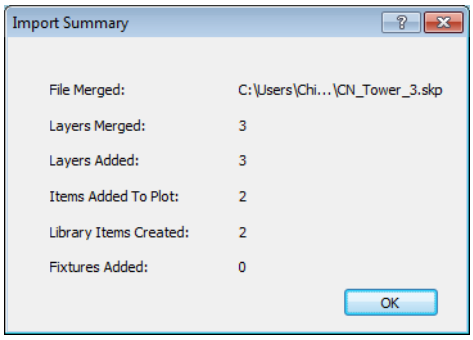


15 Select the fixture that you want to use to replace the block. If you find that you are always replacing a particular type of component with a specific fixture, you can set an option in WYSIWYG so that the substitution will automatically occur each time you import the component. For example, if you have files in which the 'PAR64M' block is always a PAR 64 fixture with an MFL bulb, you have the option of telling WYSIWYG to automatically import the component in that way. To change this option, edit the Import.lst file that is created in the Library folder (usually C:\Program Data\CAST Software\WYSIWYG\1.xx.xx.x\Library) once the first fixture is set to be remembered for future imports.

Note: WYSIWYG creates a pipe for every imported fixture since fixtures in WYSIWYG need to hang on a hang structure. If, however, fixtures are on a straight pipe represented by a straight line in the DWG file, WYSIWYG will convert the entire line into a pipe.

16 Click *Finish*.

Result: The Import Summary window appears.



17 Click *OK*.

18 If you chose *Select Basepoint with mouse* in step 7, click to place the object in the drawing.

Importing DWG/DXF files

When importing a DWG/DXF file into WYSIWYG, you have two options:

- You can *open* a DWG/DXF file.
- You can *merge* a DWG/DXF file into an existing WYSIWYG document.

If you open a DWG or DXF while another show document is currently open, you are prompted to save changes to that document before another show document is opened. Only one show document may be open at a time. When you merge documents, it enables you to add the contents of the DWG or DXF file to the contents of the current document.

3D solids contained within an AutoCAD file are automatically placed inside blocks by AutoCAD. When you import any file that contains a block, WYSIWYG will automatically explode the blocks in the DXF or DWG files for you if you decide to do this before opening them in WYSIWYG. AutoCAD files saved as Release 31 or higher generally yield the best results.

Entities that can be imported

The DWG/DXF entities that can be imported are detailed in the following table. The resulting WYSIWYG objects are also shown.

DWG/DXF entity	WYSIWYG object
Circle	Circle
Point	Point
Arc	Arc
Line	Line
Leader	Line and Text*
Ellipse	Circle or line**
2D Polyline	Line
3D Polyline	Line
Mline	Line

DWG/DXF entity	WYSIWYG object
Spline (see Note)	Spline
Text	Text
MText	Group of text items
Polyface Mesh (PFACE)	Set piece
Polygon Mesh (3D Surfaces)	Set piece
Face (3D Face)	Surface
Aligned dimension	Dimension
Region	Surface

* A leader's text component is imported as text and its line is imported as a line.

** A closed AutoCAD ellipse will be imported as a circle; otherwise, it is imported as a multi-segment line.

Entities that can be imported as a library item

- Circle
- Arc
- Line
- 2D Polyline
- 3D Polyline
- Polyline
- Mline
- Polyface Mesh (PFACE)
- Polygon Mesh (3D Surfaces)
- Face (3D Face)

Limitations

- Multilines becomes single lines.
- Multiline text becomes a group of individual text items, with each line represented by its own item.

Note: The formatting for each line is taken from that of the first character of the line, except in the case of bullets, in which case the first character after the bullet point is used.

- Segments that are not straight (for example, arcs) become straight line segments.
- A spline must contain "Fit" data for it to be imported. When a Spline is initially drawn in AutoCAD, it is defined by a series of user-provided Fit Points, which are automatically converted to Control Points by AutoCAD. To import Splines into WYSIWYG properly, you must maintain the Fit Points that define the Spline. If the Spline is defined only by Control Points, it will not import into WYSIWYG.
- Stretched (scaled) blocks are not supported.
- Blocks to be converted to library items cannot imbed other blocks.
- Blocks to be replaced by a fixture cannot imbed other blocks.

Tips

- It is recommended that you do not import 2D/3D solids whenever possible. Instead, use 3D Face and 3D Surface entities.
- It is recommended that you explode Polyface Mesh or Polygon Mesh entities in AutoCAD before importing to WYSIWYG so you can access each face in WYSIWYG.

Opening a DWG/DXF document

Before you begin

- Clean up the CAD file by removing all unnecessary layers, such as doors, windows, and architectural details. Generally you want to remove items that you do not need to see or will not use in the plot.
- Use the Purge command in AutoCAD to remove layers, blocks, and so on, that are not wanted or needed. You may want to do this several times since layers and blocks are sometimes linked to other parts of the drawing, and the Purge command might not pick them up the first time. The more unwanted items you can remove, the smaller the file size will be, and the easier/faster it is to import.

To open a DWG/DXF document

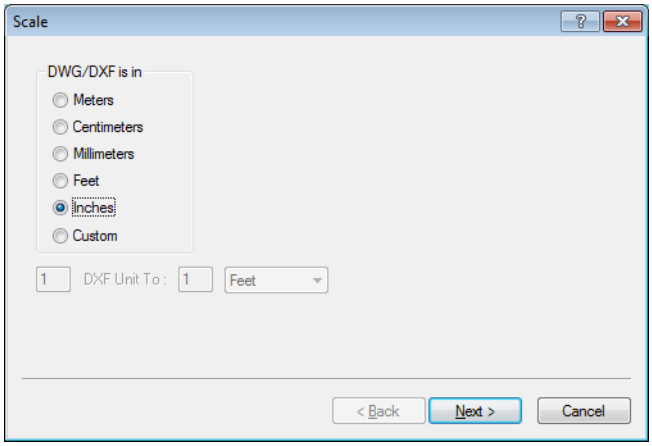
This procedure creates a new WYSIWYG document from the imported DWG/DXF file. If you are already working in a document, you are prompted to save any changes and close the document before you import the file into WYSIWYG.

1 From the **File** menu, choose **Open**.

2 From the files of type box, select the DWG or DXF type. Files of this type appear in the browser window.

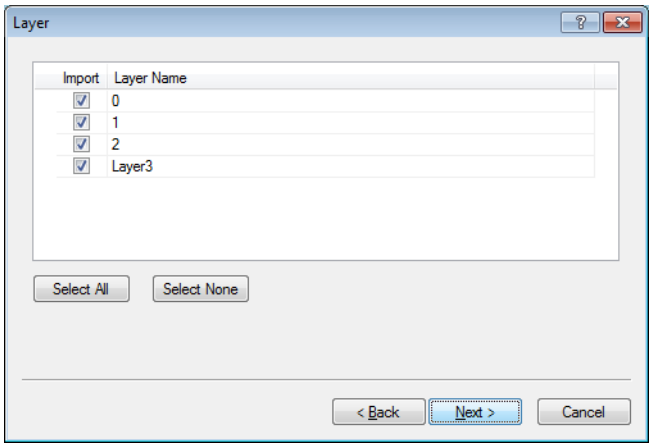
3 In the browser, click the file name, and then click *Open*.

Result: The *Scale* dialog box appears.



4 Select the unit type used in the DWG/DXF drawing.

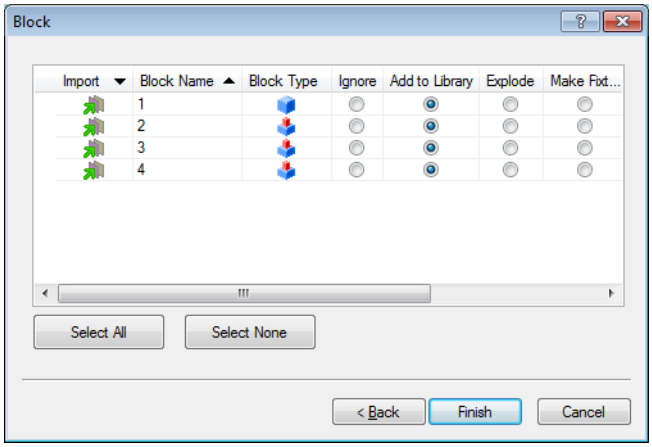
5 Click *Next*.



6 Highlight the layers that you want to import. Note that you do not need to import all the layers at once. Refer to “Importing scenario” on page 124 for suggestions on how to import the various layers. For each layer that you want to import, ensure that the *Import* check box is checked. If you do not want to import a certain layer, highlight it and deselect this checkbox.

7 Click *Next*.

Result: The Block window appears.



8 If you have blocks in your drawing, you have the option to ignore the blocks, explode the blocks, convert them into a library item, or substitute WYSIWYG fixtures for the existing blocks.

Note: 3D solids are automatically placed inside blocks by AutoCAD, and assigned an abstract block name (for example, *X1). The number of “blocks” that are found are dependent on the number of solids that were in the AutoCAD file. You will be prompted to determine what action to take for each type of “block.”

a. *Import*: This icon indicates that the block is currently on a layer being imported. This icon indicates that the block is on a layer that will *not* be imported.

b. *Block Name*: The name of the block.

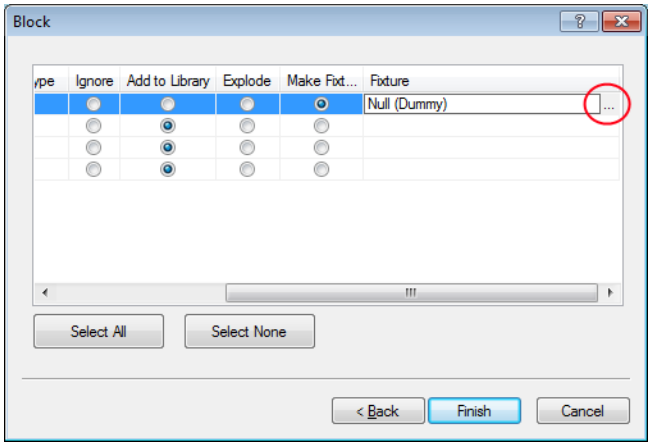
c. *Block Type*: This icon indicates that the block is a singular block and does not contain any sub-blocks. This icon indicates that the block is a nested block and contains one or many sub-blocks.

d. *Ignore*: Select this action to ignore the block. The block will not be added to the resulting file

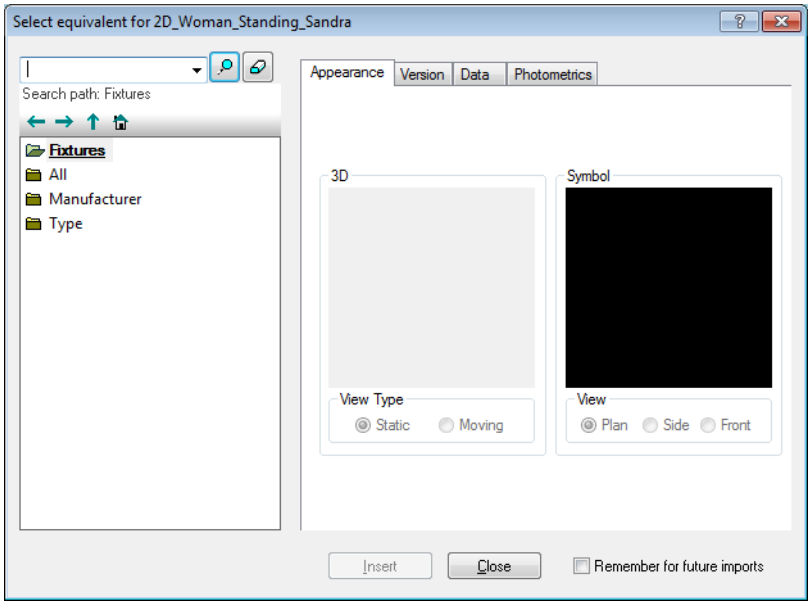
e. *Add to Library*: Converting a block into a library item imports the object and creates a duplicate of it to add to the Objects library for future use. Note that the block will only be available in the library of the current document. To make it available globally, see “To create a custom library item” on page 137.

f. *Explode*: Exploding the block does the same thing as the Explode command in AutoCAD. That is, it breaks the block into its components. You must be careful of blocks that are made up of other blocks as WYSIWYG will only explode down one level.

g. *Make Fixture*: Converting a component into a fixture replaces the blocks with a WYSIWYG fixture of your choice. Once you have selected this option, select the ellipsis in the neighboring column.



Result: The Fixture select window appears.



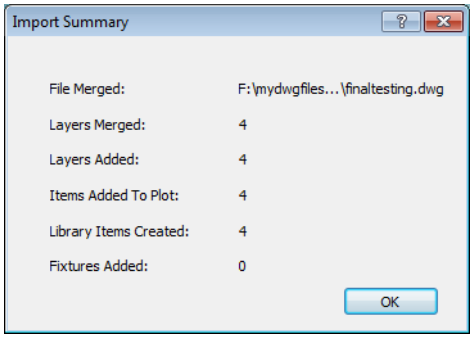
9 Select the fixture that you want to use to replace the block. If you find that you are always replacing a particular type of component with a specific fixture, you can set an option in WYSIWYG so that the substitution will automatically occur each time you import the component. For example, if you have files in which the ‘PAR64M’ block is always a PAR 64 fixture with an MFL bulb, you have the option of telling WYSIWYG to automatically import the component in that way. To change this option, edit the Import.lst file that is created in the Library folder (usually C:\Program Data\CAST Software\WYSIWYG\1.xx.xx.x\Library) once the first fixture is set to be remembered for future imports.

Note: WYSIWYG creates a pipe for every imported fixture since fixtures in WYSIWYG need to hang on a hang structure. If, however, fixtures are on a straight

pipe represented by a straight line in the DWG file, WYSIWYG will convert the entire line into a pipe.

10 Click *Finish*.

Result: The Import Summary window appears.



11 Click *OK*.

Merging a DWG/DXF document

To merge a DWG/DXF document

This procedure inserts the imported DWG/DXF file into an existing WYSIWYG document in the location of your choice.

1 From the **File** menu, choose **Merge**.

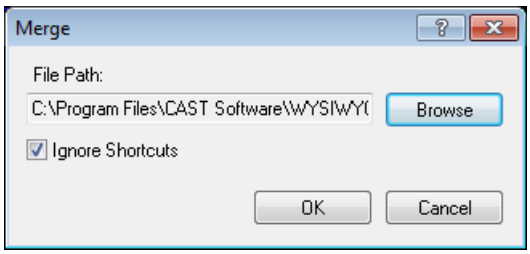
2 Click *Browse* to locate the DWG or DXF file on your computer.

3 In the Merge window that appears, beside the File name box, select the DWG or DXF file type from the drop-down list.

4 Browse to locate the file, select it, and then click *Open*.

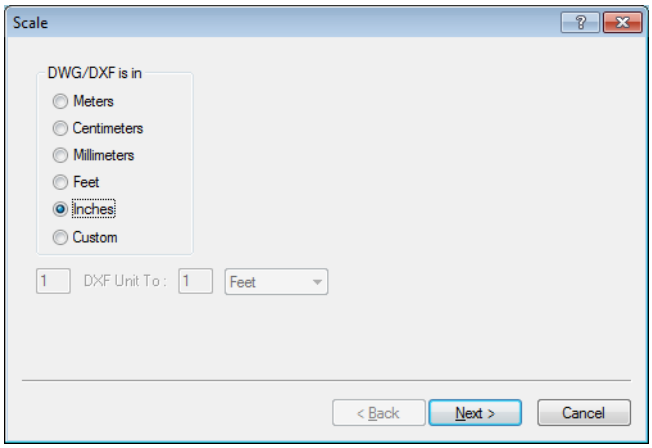
5 Leave Ignore Shortcuts selected, and then click *OK*.

Result: The *Merge* dialog box opens.



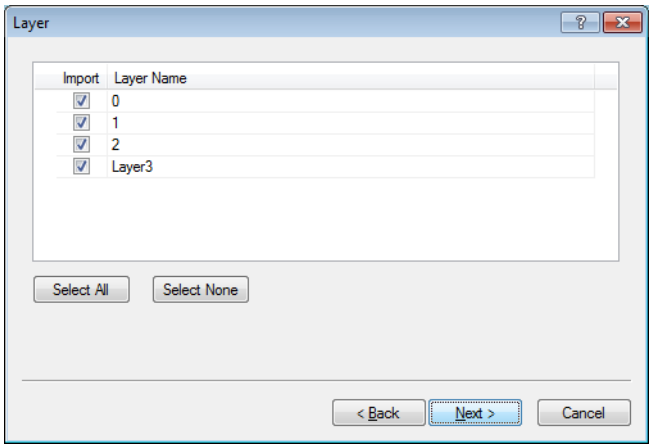
6 Click *OK*.

Result: The *Scale* dialog box appears.



7 Select the unit type used in the DWG/DXF drawing, and then click *Next*.

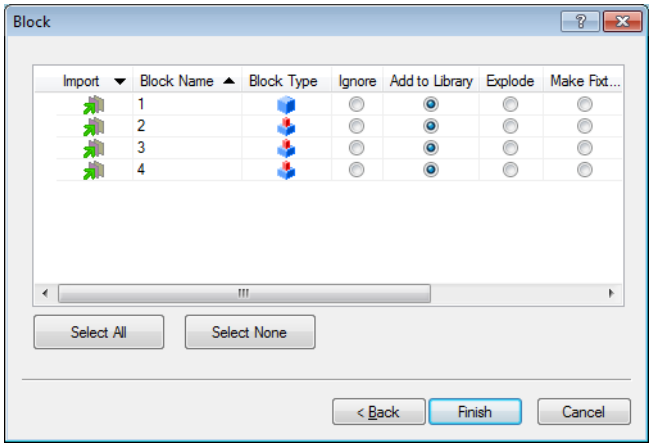
Result: The Layer window appears.



8 Highlight the layers that you want to import. Note that you do not need to import all the layers at once. Refer to “Importing scenario” on page 124 for suggestions on how to import the various layers. For each layer that you want to import, ensure that the *Import* check box is checked. If you do not want to import a certain layer, highlight it and deselect this checkbox.

9 Click *Next*.

Result: The Block window appears.



10 If you have blocks in your drawing, you have the option to ignore the blocks, explode the blocks, convert them into a library item, or substitute WYSIWYG fixtures for the existing blocks.

Note: 3D solids are automatically placed inside blocks by AutoCAD, and assigned an abstract block name (for example, *X1). The number of "blocks" that are found are dependent on the number of solids that were in the AutoCAD file. You will be prompted to determine what action to take for each type of "block."

a. *Import*: This icon indicates that the block is currently on a layer being imported. This icon indicates that the block is on a layer that will not be imported.

b. *Block Name*: The name of the block.

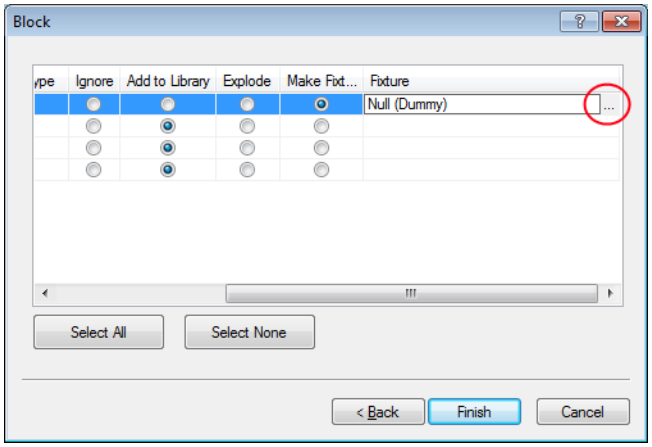
c. *Block Type*: This icon indicates that the block is a singular block and does not contain any sub-blocks. This icon indicates that the block is a nested block and contains one or many sub-blocks.

d. *Ignore*: Select this action to ignore the block. The block will not be added to the resulting file

e. *Add to Library*: Converting a block into a library item imports the object and creates a duplicate of it to add to the Objects library for future use. Note that the block will only be available in the library of the current document. To make it available globally, see "To create a custom library item" on page 137.

f. *Explode*: Exploding the block does the same thing as the Explode command in AutoCAD. That is, it breaks the block into its components. You must be careful of blocks that are made up of other blocks as WYSIWYG will only explode down one level.

g. *Make Fixture*: Converting a component into a fixture replaces the blocks with a WYSIWYG fixture of your choice. Once you have selected this option, select the ellipsis in the neighboring column.



Result: The Fixture select window appears.

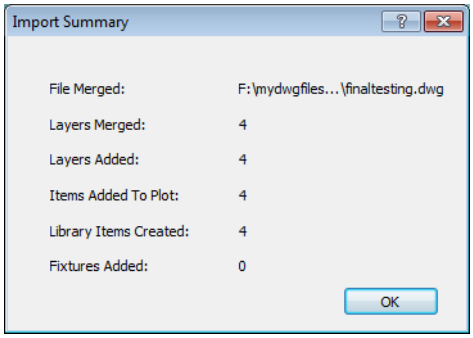
11 Select the fixture that you want to use to replace the block. If you find that you are always replacing a particular type of component with a specific fixture, you can set an option in WYSIWYG so that the substitution will automatically occur each time you import the component. For example, if you have files in which the 'PAR64M' block is always a PAR 64 fixture with an MFL bulb, you have the option of telling WYSIWYG to automatically import the component in that way. To change this option, edit the Import.lst file that is created in the Library folder (usually C:\Program Data\CAST Software\WYSIWYG\1.xx.xx.x\Library) once the first fixture is set to be remembered for future imports.

Note: WYSIWYG creates a pipe for every imported fixture since fixtures in WYSIWYG need to hang on a hang structure. If, however, fixtures are on a straight

pipe represented by a straight line in the DWG file, WYSIWYG will convert the entire line into a pipe.

12 Click *Finish*.

Result: The Import Summary window appears.



13 Click *OK*.

Importing scenario

You do not need to import everything in your file at once. You can import items one by one or separately to ensure accuracy. The following scenario may help you when importing DWG/DXF files into WYSIWYG:

1. After choosing to open a DWG/DXF file, switch to the *Layers* tab on the *Select DWG Settings* dialog box. In this tab, you can deselect the layers that you do not want imported in the first round. For example, you may only want to import the venue and all the layers related to it. In the case of a 2D CAD drawing, it is easier to extrude lines into walls and do everything else needed to complete the virtual venue. It will also be quicker as WYSIWYG will have a lot less objects with which to work.
2. Once you have completed working on the venue, you may want to merge in the hang structures if they exist in the CAD file. Using the **Merge** command from the **File** menu, you can open the same CAD file once again, and choose the appropriate layer(s) from the *Layers* tab. When WYSIWYG asks you to pick the insertion point, make sure you select *Use 0,0,0 as Basepoint*. This will ensure that everything in the new layer is placed accurately in the WYSIWYG plot.
3. Continue in the same manner with fixtures and the other objects. Note that if in the CAD file the fixtures exist on different layers, you may want to bring these in separately/one by one as well.

Exporting DWG/DXF files

If you need to send your WYSIWYG drawings to someone using another drafting program, you can export your file to the DWG or DXF file type. DWG and DXF formats are used to transfer documents to AutoCAD or other compatible drafting applications.

Entities that can be exported

2D DWG/DXF When you export WYSIWYG objects to a **2D** DWG/DXF file, all objects, including fixture attributes, are converted to lines.

3D DWG/DXF The objects that can be exported to a **3D** DWG/DXF file and the resulting DWG/DXF entities are shown in the following table. Fixture attributes cannot be exported to 3D DWG/DXF.

WYSIWYG object	DWG/DXF entity
Line	3D Polyline
Spline	Spline
Point	Point
Circle	Circle
Arc	Arc
Text	Text
Pipe	Line
Dimension	Lines and MText
All other objects	PolyFace Mesh

Note: You cannot export truss.

Limitation

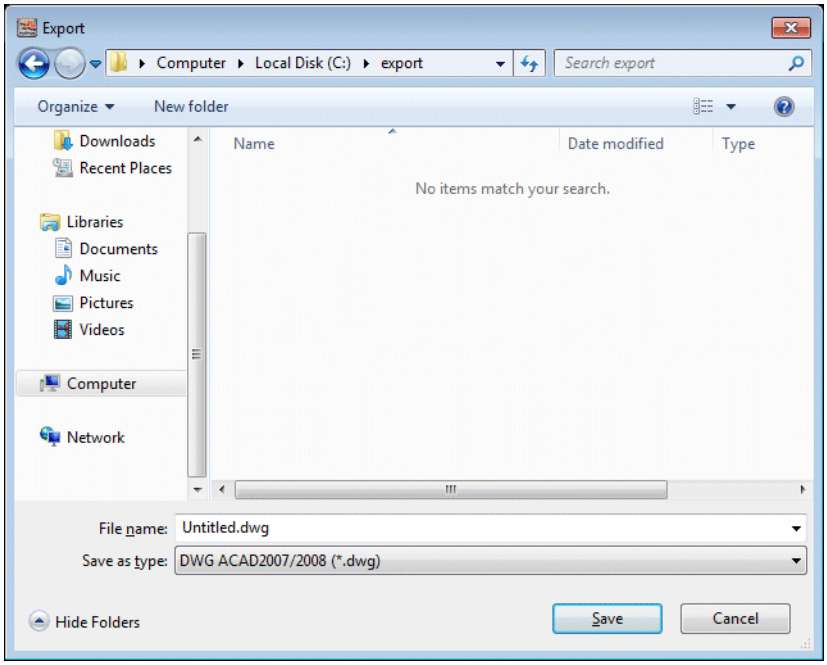
- The slope near the end of two lines may be slightly off.

To export to DWG/DXF in 2D

Note: When you export WYSIWYG objects to a **2D** DWG/DXF file, all objects, including fixture attributes, are converted to lines.

1 From the **File** menu, choose **2D DWG/DXF Export**.

2 In the *Export* dialog box, navigate to the location where you want to save the exported file. Ensure that the destination appears in the *Save in* drop-down box.

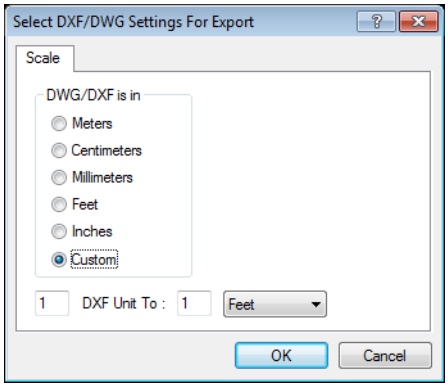


3 In the *File name* box, type the name of the exported file.

4 From the *Save as type* drop-down box, choose the exported file type. There are a number of choices available for file type, based on AutoCAD versions.

5 Click *Save*.

Result: The *Select DWG Settings* dialog box is displayed.



6 On the *Scale* tab, click the option button for the units to be used in the DWG/DXF drawing.

a. To specify a custom unit of measurement, select the *Custom* option button.

b. In the *DXF Unit To* boxes, specify the mapping of the units of measurement to use in the exported file.

7 Click *OK*.

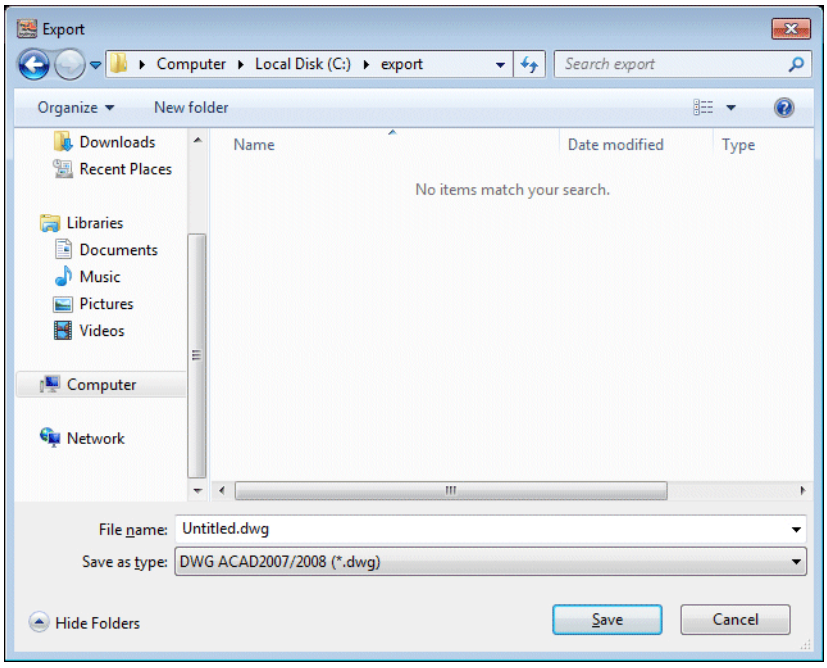
Note: The current type of the drawing will be used as the point of view in the exported 2D drawing.

To export to DWG/DXF in 3D

Note: When you export WYSIWYG objects to a **3D** DWG/DXF file, the resulting DWG/DXG entities vary. For details, see the table on page 124.

1 From the **File** menu, choose **DWG/DXF Export**.

2 In the *Export* dialog box, navigate to the location where you want to save the exported file. Ensure that the destination appears in the *Save in* drop-down box.

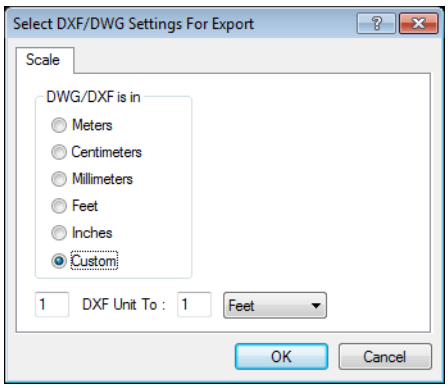


3 In the *File name* box, type the name of the exported file.

4 From the *Save as type* drop-down box, choose the exported file type. There are a number of choices available for file type, based on AutoCAD versions.

5 Click *Save*.

Result: The *Select DXF/DWG Settings for Export* dialog box appears.

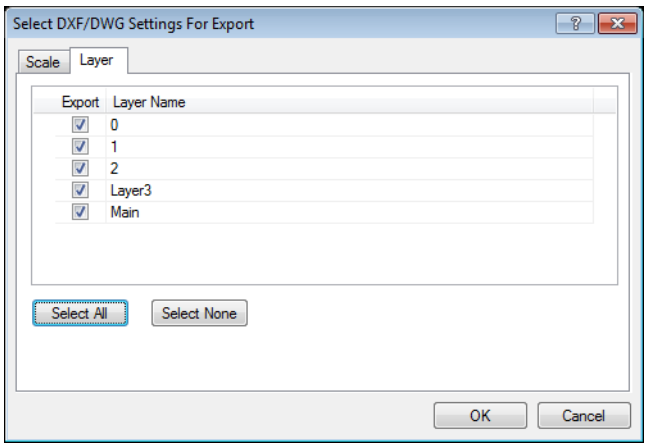


Note: You cannot export a 3D drawing from WYSIWYG Report.

6 On the *Scale* tab, click the option button for the units to be used in the DWG/DXF drawing.

- To specify a custom unit of measurement, select the *Custom* option button, and then specify the mapping of the units of measurement to use in the exported file, in the *DXF Unit To* boxes.

7 Click the *Layer* tab.



8 Select the layers that you want to include in the DWG/DXF drawing by highlighting them and ensuring that the *Export* check box is checked.

a. Click *Select All* to select all the layers that are listed.

b. Click *Select None* to deselect all layers.

9 Click *OK*.

Importing floorplans

If you have an existing floorplan saved in bitmap (.bmp), JPEG (.jpg), GIF (.gif), TIFF (.tif), PNG (.png), Targa (.tga) or Portable Document Format (.pdf), you can import it into your drawing using the Floorplan Importing Wizard, and then use WYSIWYG's Drawing Tools to "trace" it and, in so doing, create a 2D, then a 3D model of the venue in the floorplan.

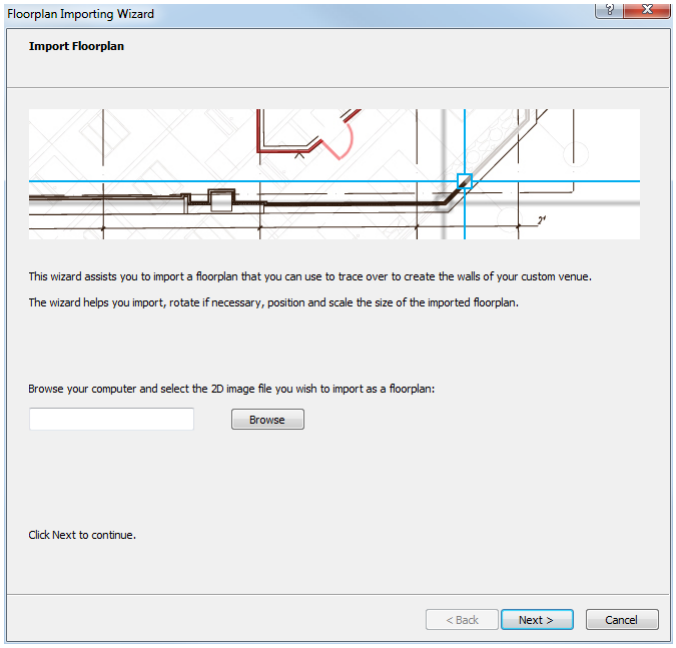
Note: You can view your imported floorplan in CAD Mode's Wireframe or Quad tabs, but only in Plan view, not in any other CAD view. It is important to note that you CANNOT print Imported Floorplans from anywhere in WYSIWYG.

To import a floorplan

Note: Before you perform this procedure, you must know the precise dimensions of at least one of the elements in your imported floorplan. For example, if it contains a stage, note the width of the stage before you import the graphic into your drawing. The floorplan should also be to scale.

1 In either the Wireframe or Quad tab, in Plan view, click **File** > **Import Floorplan...**.

Result: The Floorplan Importing Wizard window will appear.



2 In the window click *Browse*.

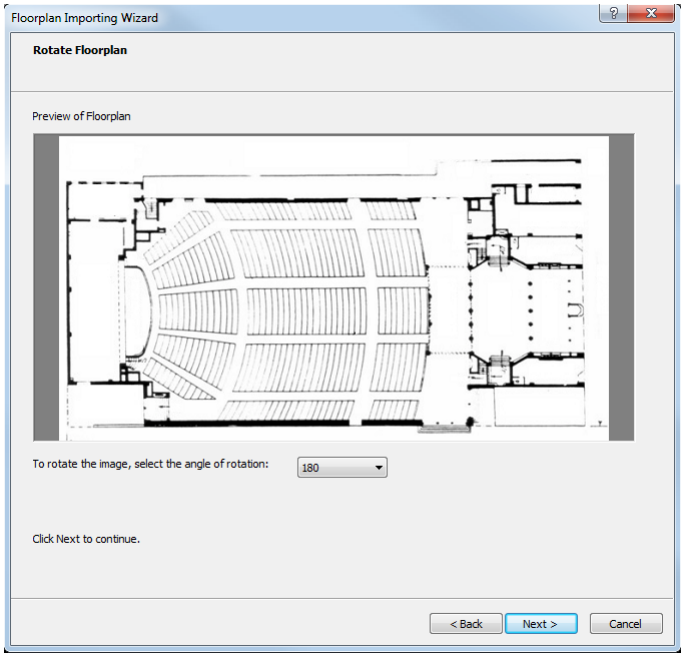
Result: The file browser window will open.

3 In the file browser window, navigate to your floorplan file, select it, and then click *Open*.

Result: The file browser window will close. The location of the floorplan image will be in the *Browse* field.

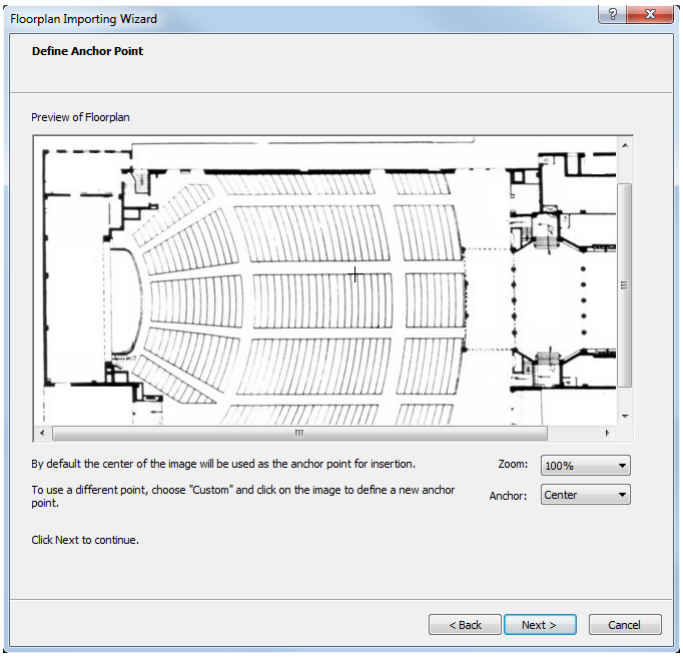
4 Click *Next*.

Result: The wizard will switch to the Rotate Floorplan page. Here the floorplan image can be rotated 0°, 90°, 180° or 270°.



5 Select the angle of rotation for the floorplan, then click *Next*.

Result: The wizard will switch to the Define Anchor Point page. You can select from predefined anchor points or create your own.

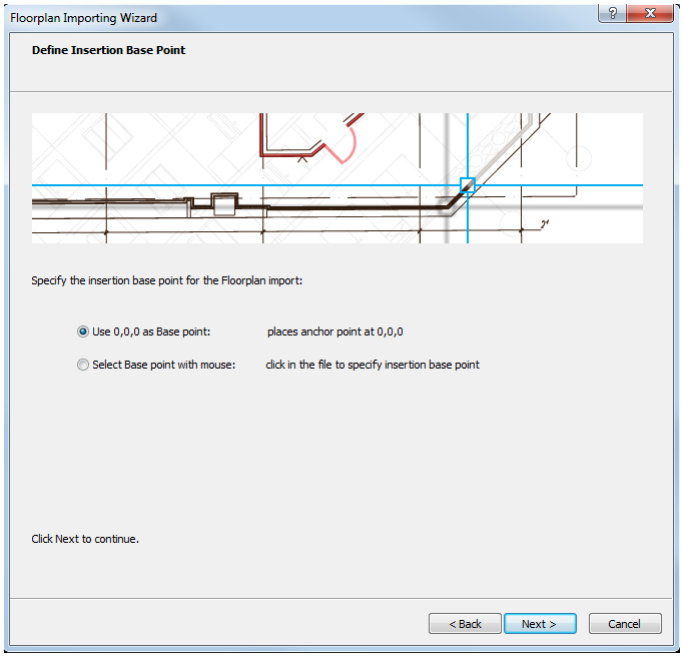


6 From the **Anchor** drop-down menu, select the anchor point of the floorplan.

a. If a custom anchor point was selected, click a point in the floor plan to create the anchor point.

7 Click *Next*.

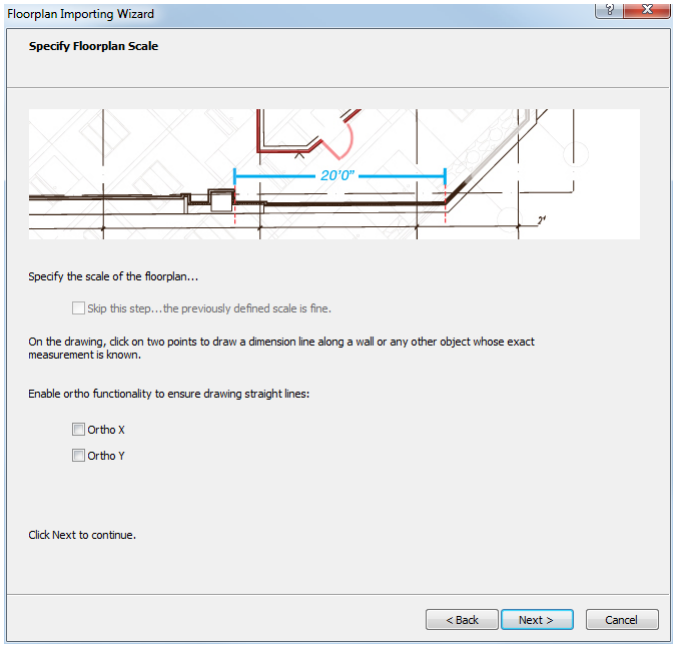
Result: The wizard will switch to the Define Insertion Base Point page. The anchor can be inserted at either the origin or a user selected point.



8 Define the insertion base point, then click *Next*.

a. If a user selected point was chosen, the wizard window will minimize. Click on a point in the WYSIWYG file where you want the insertion point to be.

Result: The wizard will switch to the Specify Floorplan Scale page. Here a dimension line is drawn between two points of the floorplan to determine its scale.



9 Ortho X and/or Ortho Y can be enabled to ensure any dimension line drawn is parallel with the respective axis.

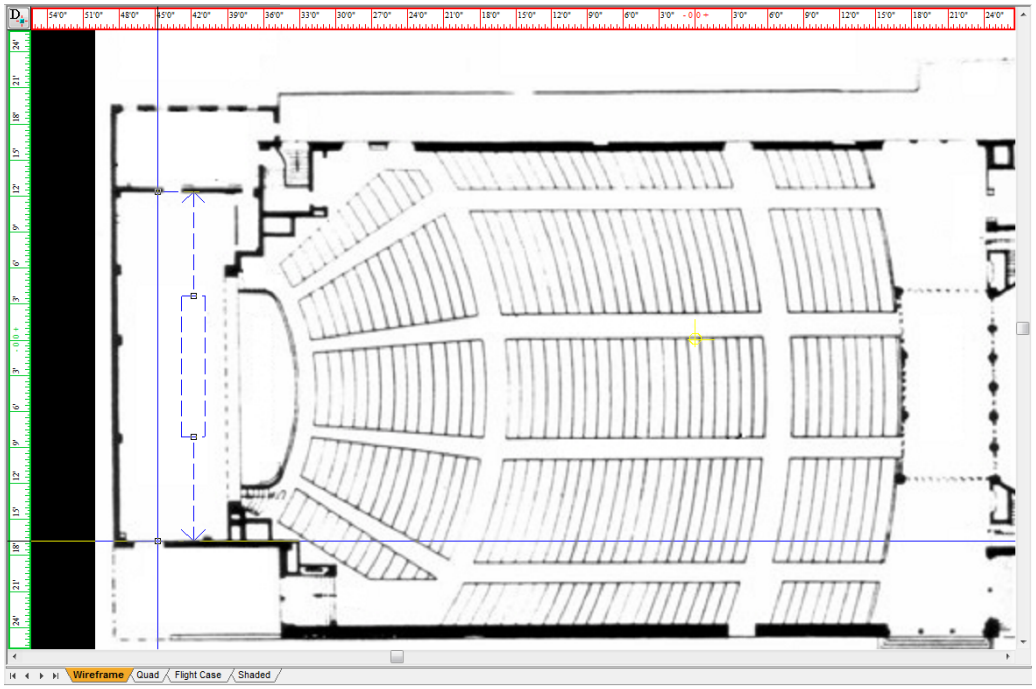
Note: This step can be skipped if you are modifying a previously imported floor plan, by marking the box next to *Skip this Step...*.

10 Click *Next*.

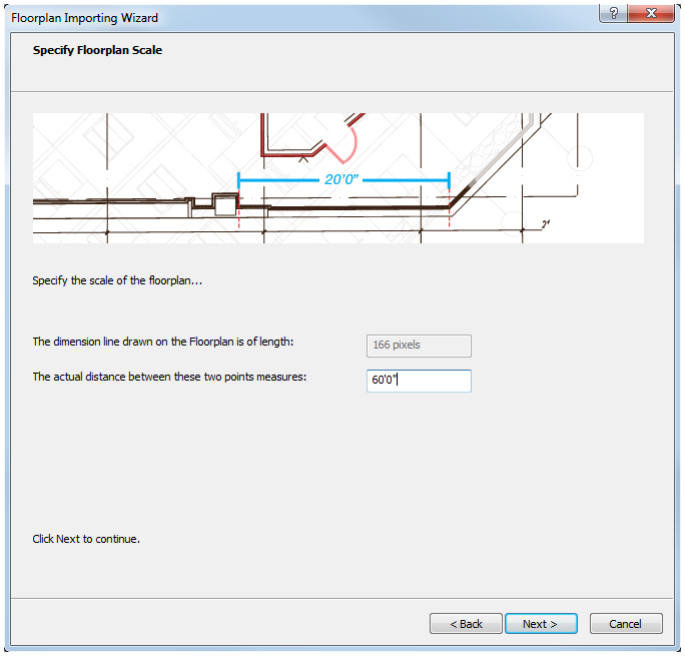
Result: The window will minimize and show the floorplan inserted into WYSIWYG.

11 In the floorplan image, choose an item in your drawing whose measurements you know, and then click on one end of the item. For example, if you know the dimensions of the stage, click on one corner of it.

12 Click on a second end of the item. For example, if you had clicked on one edge of the stage, click on an opposite edge of it.



Result: The Import Floorplan window will reappear, prompting you to enter the length of the line that you have just drawn.



13 Type the length of the line, and then click *Next*.

Result: WYSIWYG adjusts the size of the floorplan based on the scale that you have entered to ensure that your drawing is accurate. The floorplan appears in your drawing in plan view (the only view in which you can see it).

14 Click *Finish* to finish editing the floorplan. Click *Back* to change any of the previous settings.

Tip: Once you import the floorplan, you can hide it from sight by right-clicking on your drawing and selecting **Hide Floorplan**. To show the floorplan again, simply right-click and select **Show Floorplan**.

Tip: To modify the properties of the floorplan after it has been inserted, right-click the floorplan image and select **Modify Floorplan**.

Note: You can delete floorplans, but once deleted, the floorplan must be reimported in order for it to be available once again. Since this process may affect the measurements set in the above procedures, it is recommended that you only delete floorplans once your show is “finalized".

The Library Browser

Introduction

An essential component of WYSIWYG is its extensive library. You can browse the contents of the library in the Library Browser, choosing from a multitude of 3D objects to add to your plots.

Working with the Library Browser

The Library Browser is dockable, which means that it has a grab bar on the top that is used to move it around the screen. You can place it on the edges of the work area, at the top, bottom, left, or right. You can also drag it off the edge of the work area and into its own window. This window remains on top of the WYSIWYG screen and may be dragged anywhere.

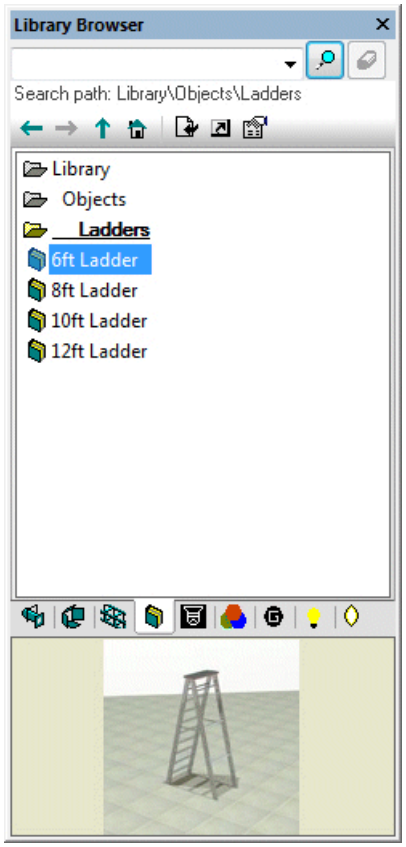
Note: If you experience difficulty undocking the browser, hold down the Ctrl key while dragging the grab bar.

To open the Library Browser

1 From the **Library** menu, choose **Browse Library**.

Tip: You can also click the *Library* on the *Draw* toolbar.

Note: You can also access the Library Browser from the **Options** menu on the Welcome window when a document is not opened.



2 Click any of the tabs along the bottom of the browser window to view the contents of different sections of the library. The library is divided into the following sections:

-  Fixtures
-  Accessories
-  Truss
-  Library Items (scenic)
-  Symbols
-  Gel (color)
-  Gobos
-  Lamps
-  Lenses

Library entries are divided into sort criteria. These criteria may be organized alphabetically, by manufacturer, by type, or by other pertinent classifications. For example, you can find the ETC Source 4 in the Fixtures section > Manufacturer > EFGH > ETC > ERS sub-menu.

Depending on the library item that you click, you may see a sample image of the item in the preview pane at the bottom of the browser.

To search the Library Browser

You can save time by using the Library Search feature. In the Library browser, simply click the category tab that you are interested in, and then type the object name (or part of it) in the *Search* box at the top of the browser panel. All objects that match your query are shown.

1 Open the Library Browser.

2 Click the category tab that you are interested in. For example, click *Fixtures*.

3 In the *Search* box, type the name of the item that you are searching for, or a partial name. For example, if you are searching for the CE Source 4 Par, you could type “par”.

4 Click the Search icon. 

Result: The results of your search appear in the browser pane.

Tip:

- To return to the top level of the current category, click the *Home* icon.
- To clear the search, but remain in the current category, click the *Clear* button. 

To view the properties of library objects

1 Open the Library Browser.

2 Navigate to and select the desired object.

3 Click the *Property* tool at the top of the Library Browser. 

4 Repeat steps 2-3 to view the properties of multiple objects. You may want to open the properties of different objects to compare data, such as photometric information.

Tips:

- You can also hold the ALT key and double-click on the object name.
- To view library objects without opening property windows, browse the library using the shortcut bar. To do this, select the shortcut bar for the type of object you want to view. Right-click in the shortcut bar, and then choose **New *<library item>***. The *Library Selection* dialog box opens with a viewing pane included.

To insert objects from the library

1 Open the Library Browser.

2 Navigate to the desired object or use the Search feature to locate it (for details, see “To search the Library Browser” on page 134).

3 Click the *Insert* tool at the top of the Library Browser. 

Tip: You can also double-click on the object name.

Note: Fixtures may only be placed on hang structures. Please refer to “Hanging and focusing fixtures” on page 274 for more information.

4 Click in your drawing to insert the object.

5 Click away from the object to deselect it.

To replace inserted library objects

You can use the Replace Library Item command to quickly and easily replace one or more library items with another.

1 Select the library item(s) that you want to replace.

2 Right-click and choose **Replace Library Item**.

3 In the Library Selection window, select the new type of library item.

4 Click *Insert*.

Result: All the library items that you selected in step 1 are replaced with the item chosen in step 3.

To create a shortcut for a library object

In the Wireframe and Quad views you can create shortcuts for frequently used objects. Shortcuts are tools found on the various shortcut bars to the left of the working area. Clicking a shortcut for an object is the same as finding the object in the Library Browser and using the *Insert* command.

1 Open the Library Browser.

2 Navigate to the desired object.

3 Click the *Create Shortcut* tool at the top of the Library Browser. 

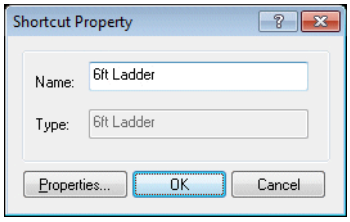
Result: A shortcut to the object will appear on the appropriate shortcut bar.

Tip: You can also right-click on the object name, and then select **Create Shortcut.**

To modify a shortcut for a library object

1 Right-click on the shortcut, and then select **Properties**.

Result: The Properties dialog box appears. An example of the accessories *Shortcut Property* dialog box is shown below.



2 In the *Name* box, type a new name for the shortcut.

The name of the fixture, color, gobo, accessories, library, or truss as listed in the library, is displayed in the *Type* box.

3 For fixtures, you can modify the type of lens and lamp associated with the fixture by selecting the appropriate option from the *Lens* and *Lamp* drop-down lists, respectively.

4 Click *Properties* to view the properties of the library entry.

5 When you are satisfied with your selections, click *OK*.

To apply a texture to a library object

You can alter the appearance of library objects by applying textures to them. You can choose from thousands of textures in the WYSIWYG library or you can apply a custom texture that you have saved as a graphic file on your computer (for example, .bmp or .jpg).

Note: Since most library objects are small with complex surfaces, it is best to apply even, plain textures to them. Complicated textures containing text or other recognizable images will not display properly on small, irregular surfaces.

1 Once you have inserted the library object in your plot, click on it to select it, and then right-click and select **Properties**.

2 Click the *Appearance* tab.

3 In the left pane, highlight the elements of the library item to which you want to apply the texture, and then click the type of texture file that you want to use, either from the Library or a custom texture that you have saved. For detailed descriptions of the texture and material options available on the *Appearance* tab, see “Appearance tab” on page 196.

4 Click *OK* when you are finished.

Custom library items

You can create custom library items, custom conventional fixtures, custom gobos, and custom color and gobo wheels or scrolls.

When creating custom gobos and custom library items, you are given the option to make the item or gobo available to other documents, in which case the item or gobo is saved to the application’s library. If you do not choose to make the items available to other documents, the items are only saved within the one document. In either case, the items are saved and visible in the document regardless of which computer (or application) the document is opened in.

Custom conventional fixtures must be created from the WYSIWYG welcome screen and are saved directly to your application's library. The new conventional fixture is available for you to use in all documents created using this application. If used in a show document, the custom conventional fixture is available in the document regardless of which computer (or application) the document is opened in.

Custom color, gobo wheels, and scrolls are only saved in the document and cannot be saved to the application's library. They travel with the document, but if you need to use the same wheel or scroll on another show, you must regenerate it every time.

Note: You can delete custom library items, custom conventional fixture, and custom gobos. For details, see "To delete a custom library item" on page 176, "To delete custom fixtures" on page 279, and "To delete custom gobos" on page 320.

To create a custom library item

See "To create a custom library item" on page 176.

To create a custom fixture

See "To create a custom fixture" on page 278.

To create custom color or gobo lists

See "To create a custom color/gobo list" on page 317.

To create a custom gobo

See "To create a custom gobo" on page 320.



Drawing objects

Introduction

In WYSIWYG, the **Draw** menu lists the objects that you can draw. Keep the following tips and aids in mind when you are drawing the various objects.

Drawing tips

- At any time, instead of clicking points with the mouse, you can type in the desired coordinates. When you start typing, the Command Line automatically appears in the lower left-hand corner of the window.
- Create shortcuts for library objects.
- Use the missing coordinate.

Drawing aids

Several commands and icons are available to help you when drawing in CAD mode. For a complete list of these aids, refer to “Keyboard shortcuts and hot keys” on page 561.

Drawing a custom room

The Room Builder tool allows you to build a custom room of any shape in *Wireframe* view.

To draw a custom room

1 From the **Draw** menu, choose **Room Builder**.

Result: A cross-hair will appear on your cursor in the drawing wireframe.

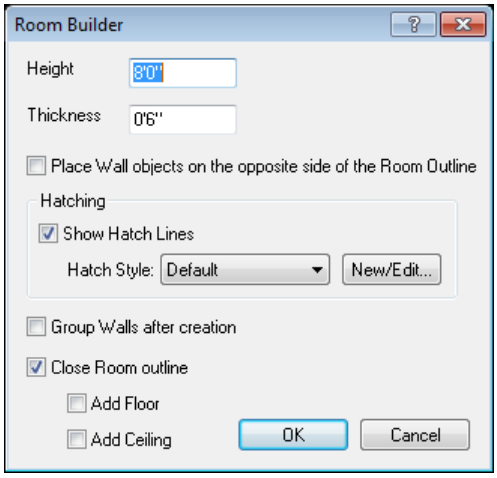
2 Click to place a corner of the wall at the location of the cursor. Each subsequent click will connect the current corner to the previous corner.

3 To finish the room drawing, connected the first and last created corner.

Note: Alternately, the room can be closed, connected the first and last created corner automatically, by right clicking and selecting **Close Room Outline**.

Note: To finish a room without closing it, right click and select **Finish Room Outline.**

Result: The *Room Builder* window will appear.



4 In the *Room Builder* window, enter in the missing characteristics of the room walls.

5 Click the *Invert Wall* check box if you want to place object on the opposite side of the Room Outline.

6 Click the *Group Walls after creation* check box if you want all created walls to be in the same group.

7 Click the *Close Room* check box if you want to add floors or ceilings to the room.

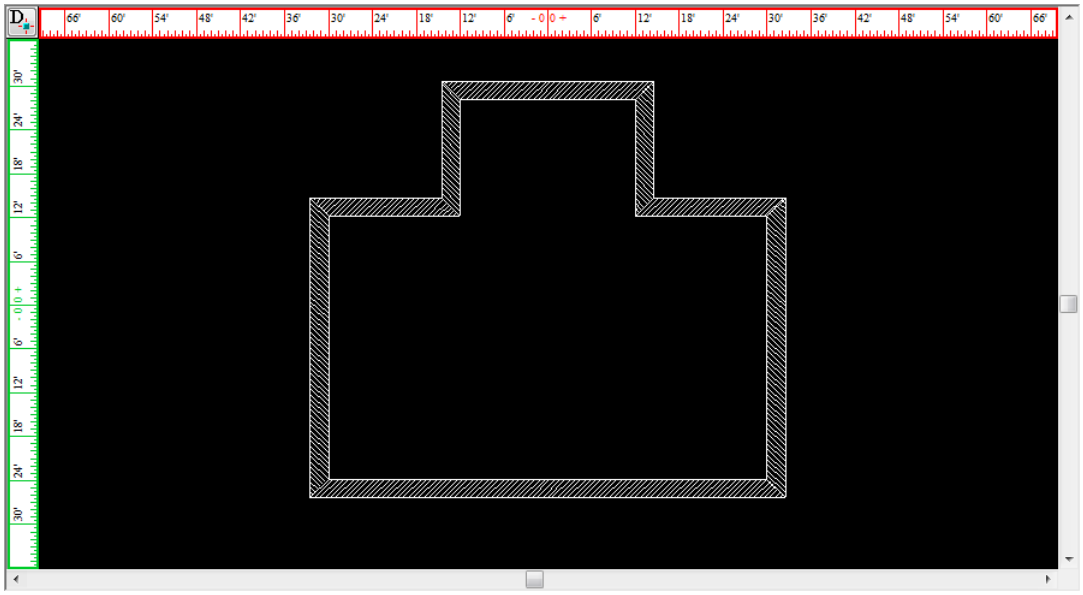
Note: This option is only available if the room is closed. When a room is closed, WYSIWYG automatically makes the Walls transparent from outside of the room looking inwards, and makes the ceiling and floor single sided, so when the camera is outside of the room the user can see inside. These properties can be changed if the user wishes, by going to the objects' properties

a. Click the *Floor* check box to add a floor to the room

b. Click the *Ceiling* check box to add a ceiling to the room.

8 Click *Enter*.

Result: The custom room will be created in *Wireframe* view.



Drawing a venue

To begin your drawing, insert a venue.

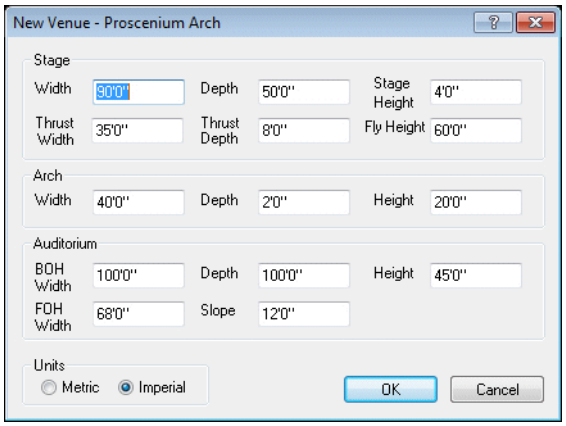
To insert a venue

1 From the **Draw** menu, choose **Venue**.

Result: A sub-menu will appear prompting you to choose the type of venue to be inserted.

2 Select **Black Box**, **Arena**, **Proscenium Arch**, **Frame Tent**, or **Circus Tent**.

Result: A dialog appears in which you can modify the properties of the room. The *Proscenium Arc*h dialog box is shown below.



3 Accept the defaults, or choose custom properties.

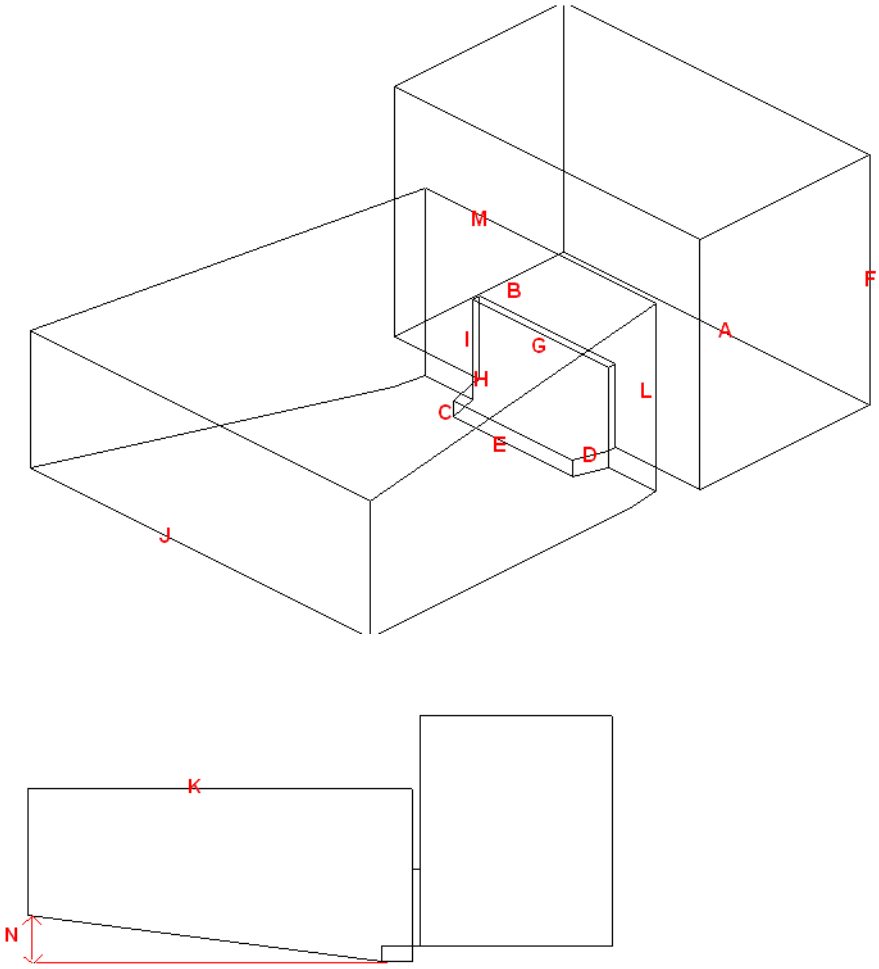
4 Click *OK*.

Result: The selected venue is inserted into the drawing.

The venue types are defined as follows:

Proscenium Arch

A proscenium arch venue is a basic theatrical space with a stage, proscenium arch, and an auditorium. The origin is set at the intersection of the center line and the proscenium line on the stage floor.



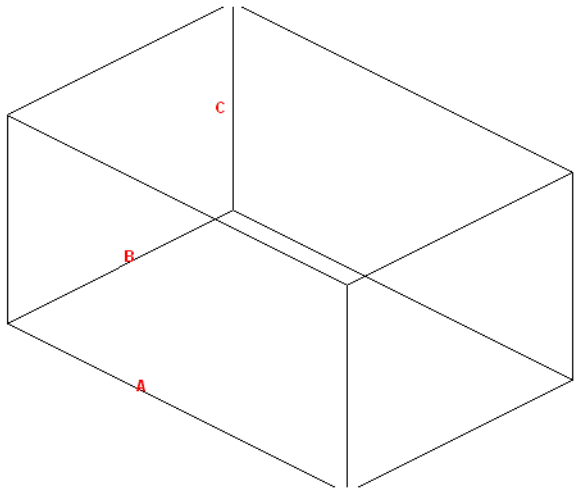
The following entries describe the parts of the proscenium arch.

- A: The stage width; the dimension from the extremity of the SR wing to the extremity of the SL wing. In WYSIWYG, the default value is 90’0”.
- B: The stage depth; the dimension from the proscenium line to the back of the stage space. In WYSIWYG, the default value is 50’0”.
- C: The stage height; the dimension from the auditorium floor to the stage deck. In WYSIWYG, the default value is 4’0”.
- D: The thrust depth; the dimension from the edge of the stage to the proscenium line. In WYSIWYG, the default value is 8’0”.
- E: The thrust width; the dimension from the extremity of the SR edge of the thrust to the SL extremity of the thrust. This dimension cannot be greater than the Arch width (G). In WYSIWYG, the default value is 35’0”.
- F: The Fly height; the dimension from the stage deck to the top of the fly house. In WYSIWYG, the default value is 60’0”.
- G: The proscenium arch width. In WYSIWYG, the default value is 40’0”.
- H: The depth of the proscenium arch wall. In WYSIWYG, the default value is 2’0”.
- I: The height of the proscenium arch; the dimension of the opening of the proscenium arch. In WYSIWYG, the default value is 20’0”.

- J: The back of house (BOH) width; the dimension of the auditorium from the extremity of house right to the extremity of house left at the back of the house. In WYSIWYG, the default value is 100’0”.
- K: The auditorium depth; the dimension from the proscenium line to the back of the house. In WYSIWYG, the default value is 100’0”.
- L: The auditorium height; the dimension from the ceiling of the auditorium to the floor of the auditorium at the edge of the stage. In WYSIWYG, the default value is 45’0”.
- M: The front of house (FOH) width; the dimension of the auditorium from the extremity of house right to the extremity of house left at the front of the house. In WYSIWYG, the default value is 68’0”.
- N: The auditorium slope height. In WYSIWYG, the default value is 12’0”.

Black Box

A black box venue is an empty rectangular room. The origin is set at the center of the room on the floor.

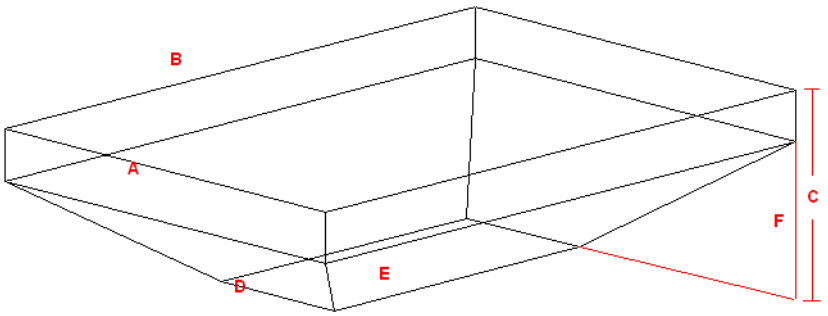


The following entries describe the parts of the black box:

- A: The width of the room. In WYSIWYG, the default value is 60’0”.
- B: The depth of the room. In WYSIWYG, the default value is 40’0”.
- C: The height of the room. In WYSIWYG, the default value is 30’0”.

Arena

An arena venue is a basic stadium space. The following entries describe the parts of the arena. The origin is set at the center of the venue on the floor or ice rink.



- A: The width of the arena. In WYSIWYG, the default value is 170’0”.

- B: The depth of the arena. In WYSIWYG, the default value is 250’0”.
- C: The total height of the arena. In WYSIWYG, the default value is 60’0”.
- D: The width of the floor or ice rink. In WYSIWYG, the default value is 60’0”.
- E: The depth of the floor or ice rink. In WYSIWYG, the default value is 130’0”.
- F: The height of the stands or seating; the portion of the total height that is occupied by seating or stands. In WYSIWYG, the default value is 40’0”.

Tent

There are two types of tents that you can create in WYSIWYG—frame tents and circus tents.

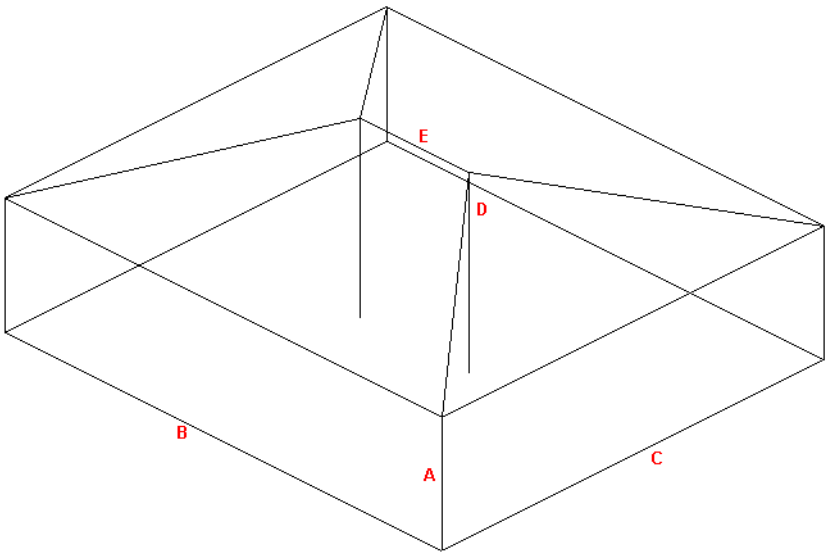
- **Frame tents** These tents are square or rectangular in shape and usually have two center poles. Each wall is made of a single piece of fabric supported by as many poles as you specify.
- **Circus tents** These tents are styled after the classical “Big Top” of the circus. They are more rounded in shape than the frame tent, the walls comprising multiple sections, each supported by as many poles as you specify. The roof is held up by one center pole.

When drawing either type of tent, you can specify many different properties, including the number of poles, length, height, or perimeter of the walls, and the color of different parts of the tent.

You can also choose from different materials and you can choose whether you want to apply either a custom color or texture to different parts of the tent. When assigning a texture, you can choose your own texture, such as a company logo or symbol, or you can choose a texture from the WYSIWYG library.

The origin for both types of tents is at the center of the tent on the floor.

Frame tent

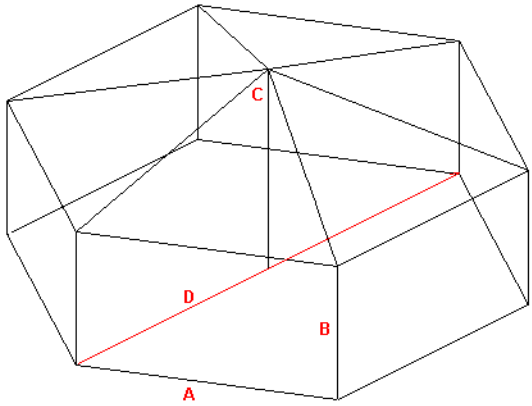


The following entries describe the parts of the frame tent:

	Description	New Venue - Tent Field	WYSIWYG Default Value
A	The height of the tent walls.	Wall height	20’0”

	Description	New Venue - Tent Field	WYSIWYG Default Value
B	The length of the tent walls.	Width	80’0”
C	The width the tent walls.	Depth	70’0”
D	The height that the center pole extends above the height of the walls.	Height above walls	10’0”
E	The distance between the center poles.	Separation	20’0”

Circus tent



The following entries describe the parts of the circus tent:

	Description	New Venue - Tent Field	WYSIWYG Default Value
A	The number of individual segments in the tent.	Segments	6
B	The height of the tent walls.	Wall height	20’0”
C	The height that the center pole extends above the height of the walls.	Height above walls	10’0”
D	The diameter of the tent, measured from pole to opposite pole.	Diameter	70’0”

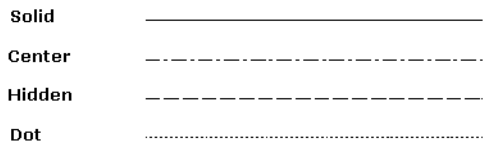
Drawing lines

Lines are 2D objects. Lines are drawn continuously, allowing you to create shapes made up of multiple vertices.

Lines come in four styles:

- Solid
- Center
- Hidden
- Dot

Line styles determine how line objects appear on your plot and can be modified at any time. They look similar to the following.



You can make a multi-vertex line appear as a spline or french curve by selecting the option in the properties box. You can then drag the markers around to adjust the curves. Alternatively, you can choose to draw a spline directly. Splines have the same line style options as regular lines.

Lines can be extruded into surfaces. For more on extruding lines, refer to “Extrude” on page 233.

To draw a line

1 From the **Draw** menu, choose **Line**.

2 From the sub-menu, select **Solid**, **Dot**, **Center**, **Dashed**, or **Spline**.

Tip: You can also use the appropriate line tool on the *Draw* toolbar. The available line tools are as follows:



- Solid
- Dot
- Center
- Dashed
- Spline

3 Click on the drawing at the starting point of the line.

4 Drag the next vertex to its end point and click.

5 Continue to place vertices of the line as needed.

6 To end the line at its last end point, right-click and choose **Finish Line**. To abort the line entirely, choose **Abort Line**. This erases the whole line from the drawing.

Drawing points

Points are identifiers of a specific coordinate in 3D space. Points are inserted as references or as scenic elements.

To draw a point

1 From the **Draw** menu, choose **Point**.

Tip: You can also use the *Point* tool on the *Draw* toolbar. 

2 Click on the drawing to place the point.

Note: Points can be drawn as a dot, a cross, a square, or a circle.

To set the point type default

1 From the **Options** menu, choose **Show Options**.

2 Click the *Object Defaults* tab.

3 Select the desired point type.

4 Click *OK*.

Drawing circles

There are two ways in which you can draw circles:

- non-interactive mode in which you type the exact horizontal and vertical radius of the circle
- interactive mode in which you click and drag to set the radius

To draw a circle in non-interactive mode

1 From the **Draw** menu, choose **Circle**.

Tip: You can also use the *Circle* tool on the *Draw* toolbar. 

Result: The New Circle dialog box appears.

2 In the *Horizontal Radius* box, type the horizontal radius for the circle. The default value is 4’0”.

3 In the *Vertical Radius* box, type the vertical radius for the circle. The default value is 4’0”.

Note: To ensure that the circle remains proportionately the same when resized, leave the *Lock ratio* check box checked. If you deselect this check box, then you can manually resize the circle in any direction, regardless of its original measurements.

Note: To convert the circle into a surface, mark the check box by **Convert to Surface**.

4 Click *OK*.

5 Click on the drawing to place the circle.

6 Click elsewhere in the drawing to finish placing the new circle.

To draw a circle in interactive mode

Note: To turn interactive mode on, click the Interactive Mode button on the toolbar. 

1 From the **Draw** menu, choose **Circle**.

Tip: You can also use the *Circle* tool on the *Draw* toolbar. 

2 Click to place the center of the circle.

3 Click to set the radius for the circle.

Drawing arcs

There are two ways in which you can draw arcs:

- non-interactive mode in which you type the exact radius, start, and end angles of the arc
- interactive mode in which you click and drag to set the arc

To draw an arc in non-interactive mode

1 From the **Draw** menu, choose **Arc.**

Tip: You can also use the *Arc* tool on the *Draw* toolbar. 

2 In the *Radius* box, type a radius for the arc. The default value is 4’0”.

3 In the *Start Angle* box, type the start angle. The default value is 0.00.

4 In the *End Angle* box, type the end angle for the arc. The default value is 180.00.

5 Click *OK*.

6 Click on the drawing to place the arc.

7 Click elsewhere in the drawing to finish placing the arc.

To draw an arc in interactive mode

Note: To turn interactive mode on, click the Interactive Mode button on the toolbar. 

1 From the **Draw** menu, choose **Arc**.

Tip: You can also use the *Arc* tool on the *Draw* toolbar. 

2 Click to place the start point of the arc.

3 Click to place the end point of the arc.

4 Click to place the middle point of the arc.

5 Click elsewhere in the drawing to finish placing the arc.

Drawing elliptical arcs

There are two ways to draw an arc:

- non-interactive mode in which you specify the elliptical arc radius, start and end angles and place the arc in the drawing.
- interactive mode in which you draw the elliptical arc.

To draw an elliptical arc in non-interactive mode

1 From the **Draw** menu, choose **Elliptical Arc**.

Tip: You can also use the *Arc Elliptical* tool on the *Draw* toolbar. 

Result: The *New Elliptical Arc* dialog box opens.

2 In the *Radius* box, enter a radius for the elliptical arc. The default value is 4’0”.

3 In the *Start Angle* box, enter the start elliptical angle. The default value is 0.00.

4 In the *End Angle* box, enter the end angle for the elliptical arc. The default value is 180.00.

5 Click *OK*.

6 Click on the drawing to place the elliptical arc.

7 Click and drag the mid point of the arc to create the desired elliptical

To draw an arc in interactive mode

Tip: To turn interactive mode on, click the Interactive Mode button on the toolbar. 

1 From the **Draw** menu, choose **Elliptical Arc**.

Tip: You can also use the *Arc Elliptical* tool on the *Draw* toolbar. 

2 Click to place the start point of the elliptical arc.

3 Click to set the horizontal and vertical radii of the elliptical arc.

4 Click to set the start point of the ellipse.

5 Click to set the end point of the ellipse.

Drawing rectangles

There are two ways to draw rectangles:

- non-interactive mode in which you enter the exact length and width of the rectangle.
- interactive mode in which you draw the size of the rectangle.

To draw a Rectangle in non-interactive mode

1 From the **Draw** menu, choose **Rectangle**.

Tip: You can also use the *Rectangle* tool on the *Draw* toolbar. 

Result: The *New Rectangle* dialog box opens.

2 In the *Length* box, type the length for the rectangle. The default value is 4’0”.

3 In the *Width* box, type the width for the rectangle. The default value is 8’0”.

Note: To ensure that the rectangle remains proportionately the same when resized, check the *Lock ratio* check box. Otherwise you can manually resize the rectangle in any direction, regardless of its original measurements.

Note: To convert the rectangle into a surface, mark the check box by **Convert to Surface**.

4 Click *OK*.

5 Click in the drawing to place the rectangle.

To draw a Rectangle in interactive mode

Note: To turn interactive mode on, click the Interactive Mode button on the toolbar. 

1 From the **Draw** menu, choose **Rectangle**.

Tip: You can also use the *Rectangle* tool on the *Draw* toolbar. 

Result: The *New Rectangle* dialog box opens.

2 Click to place one corner of the rectangle.

3 Click to place the opposite corner for the rectangle.

Drawing text labels

Before you draw your text labels in CAD mode, you can select the font that will be used in all labels. You can choose from any font installed on your computer.

To change the Text Label font globally

You can specify the font that you would like to appear globally in all text labels, both new and existing.

1 In any drawing mode, click **Options > User Options**.

2 Click the *Font Styles* tab.

3 Under CAD Text Labels, from the *Font* drop-down list, select the font that you want to appear in all new text labels.

4 Choose whether you want the letters to appear bold, underlined, or in italics.

5 Click *OK*.

To draw a text label

1 From the **Draw** menu, choose **Text Label**.

Tip: You can also use the *Text Label* tool on the *Draw* toolbar. 

2 In the *Text* box, type the desired text.

3 Type a height for the text. This is similar to selecting a font size. The default value is 2'0".

4 Click on the drawing to place the text label.

Note:

- To change the font of existing text labels in CAD mode, see "To change the font of Text Labels" below.
- The insertion point for the text label is at the intersection of the crosshairs.

To change the font of Text Labels

To change the font and appearance (bold, italic, or underline) of existing labels, you can use the label's Properties window or the Text toolbar. The following procedure outlines how to use the Properties window.

1 Select the text label whose font you want to change.

2 Right-click, and then choose **Properties**.

3 Click the *Text Label* tab.

4 Under Font, deselect the check mark beside *Use Document Defaults*.

5 From the *Font* drop-down list, select the new font.

6 Choose whether you want the letters to appear bold, underlined, or in italics.

7 Click *OK*.

To set alignment for a text label

You can use the alignment buttons on the Text toolbar to align text labels.



1 Select the text label that you want to align.

2 On the Text toolbar, click the appropriate button to align the text label in the desired position (left, center, right, top, middle, or bottom).

Result: The text label is relocated around the insertion point, based on the options selected.

Drawing dimension objects

Dimension lines are 2D objects. They have a start and end point, and measure and display the distance covered based on the measurement mode selected. The measurement modes available are: X, Y, XY, XZ, YZ, and XYZ. The dimension will only be visible in one view type. The view type is set when the dimension is drawn and is dependent on the plot type and workplane in which the dimension is drawn.

There are several types of dimensions available:

- Linear dimensions: These dimensions measure and display the distance covered between two points in your plot, based on the measurement mode selected.
- Arc length dimensions: These dimensions are used to measure the length of arcs drawn in CAD mode.
- Radial dimensions: These dimensions are used to measure the radius of any circle or arc drawn in CAD mode.

You can also use the Protractor tool to define precise angles in your plot, which comes in handy for verifying angles and helping out with the drawing process.

Tip: The properties of dimension objects, such as the arrow type, can be changed in the **Dimensions Styles** tab under **Options > User Options....**

To draw a linear dimension

1 From the **Draw** menu, **Dimensions** sub-menu, choose **Linear Dimension**.

Tip: You can also click the *Linear Dimension* tool on the *Draw* toolbar. 

2 Click on the drawing to set the dimension start point.

3 Right-click to set the measurement mode.

4 Click on the drawing to set the dimension end point.

5 Click and drag the grab point in the center of the dimension text to drag the text to the correct side of the object, if required.

6 Click to set the dimension line.

Tips:

- Use snaps to connect dimensions directly to other objects.
- To adjust the font used in all dimensions, click **Options** > **User Options** > **Font Styles**. In the CAD Dimensions area, select the font options to be used in all dimensions.

Arc Length Dimensions

The arc length can be displayed for any arc drawn in CAD mode. Once enabled, the arc length dimension is attached to the arc and automatically updates itself whenever the arc is changed. The dimension is only visible in the view type in which it was drawn.

Although the arc length dimension is attached to the arc, it is a separate entity. Each arc length is individually selectable and has its own properties. Arc length dimensions can be separated onto their own layers, and given their own colors and line weights.

To draw an arc length dimension

1 In a CAD wireframe, select the arc for which you want to display an arc length dimension.

2 From the **Draw** menu, **Dimensions** sub-menu, select **Arc Length Dimension**.

Tip:

- To adjust the font used in all dimensions and with the Protractor tool, click **Options > User Options > Font Styles**. In the CAD Dimensions area, select the font options to be used in all dimensions and with the Protractor tool.

Radial Dimensions

The radius of a circle/arc is defined as the distance from the center of a circle/arc to its perimeter. The radius can be displayed for any circle or arc drawn in CAD mode. Once enabled, the radial dimension attaches to the circle/arc and automatically updates itself whenever the circle/arc is changed. Note that the dimension is only visible in the view type in which it was drawn.

Although the radial dimension is attached to the arc, it is a separate entity with its own properties and can be individually selected. Radial dimensions can be separated onto their own layers, and given their own colors and line weights.

To draw a radial dimension

1 In a CAD wireframe, select the circle/arc for which you want to display a radial dimension.

2 From the **Draw** menu, **Dimensions** sub-menu, select **Radial Dimension**.

Tip:

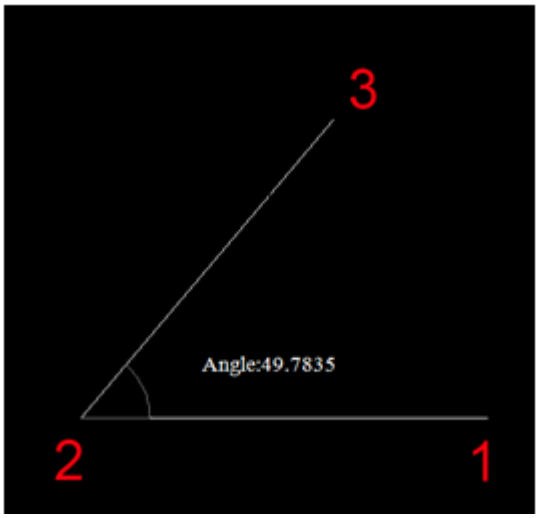
- To adjust the font used in all dimensions and with the Protractor tool, click **Options > User Options > Font Styles**. In the CAD Dimensions area, select the font options to be used in all dimensions and with the Protractor tool.

Protractor Tool

The protractor tool lets you define precise angles in your plot, which helps when verifying angular measurements, and can act as a guide for the drawing process.

To use the Protractor tool

1 From the **Draw** menu, **Dimensions** sub-menu, select **Protractor Tool**.



2 Click once to define the vertex of the protractor (this is the point at which the two lines of your angle meet, shown as point number 2 in the diagram above).

3 Click a second time to define the first end point (point number 1 in the diagram above).

4 Click a third time to define the second end point (point number 3 in the diagram above).

Result: The new object displays the interior (or exterior) angle defined by the two lines.

Tips:

- You can change the Protractor option of measuring the exterior or interior angle by toggling the *Flip Angle* check box in the Protractor Properties page. Simply select the protractor, right-click and choose **Properties > Protractor tab**.
- To adjust the font used in all dimensions and with the Protractor tool, click **Options > User Options > Font Styles**. In the CAD Dimensions area, select the font options to be used in all dimensions and with the Protractor tool.

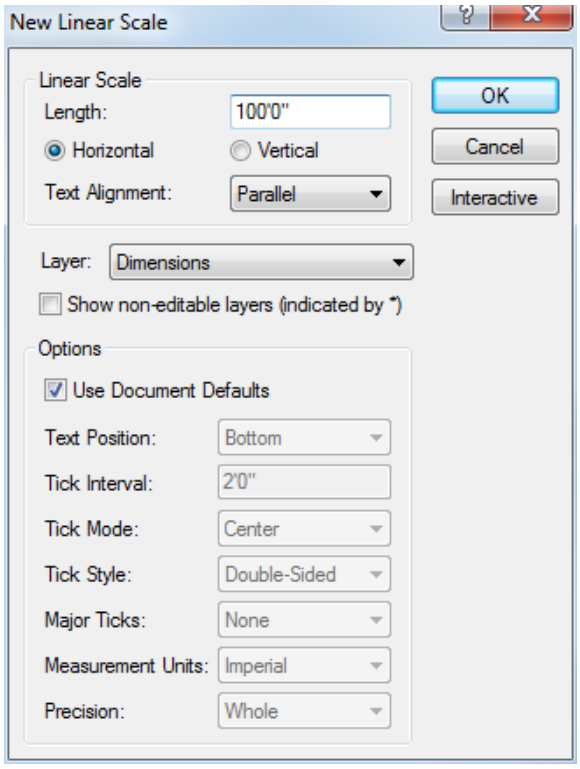
Linear Scales

A linear scale is a visible line divided into equal proportions used to compare distances in WYSIWYG wireframe to actual distances.

To draw a linear scale

1 From the **Draw** menu, **Dimensions** sub-menu, select **Linear Scale**.

Result: The *New Linear Scale* window will appear.



2 In the *Length* field, enter in the desired length of the linear scale.

3 Select either *Horizontal* or *Vertical* to determine how the line scale is positioned.

4 From the *Text Alignment* drop down menu, choose how the linear scale text will be presented.

5 From the Layer drop down menu, choose in which layer the linear scale will be visible.

6 To change the default document settings of the linear scale, unmark the check box next to *Use Document Defaults*.

7 Click *OK*.

Result: The linear scale is attached to the cursor.

8 Click to place the linear scale in the drawing.

Drawing surfaces

Surfaces are 2D objects. Surfaces are drawn continuously, stretching out behind the cursor and can be used to create backdrops or other flat objects. Surfaces can be extruded to create odd shaped risers, or other custom shaped 3D objects. For more information on extruding, refer to “Extrude” on page 233.

To draw a surface

To ensure that textures, if any, are applied properly, the surface must be drawn in a counter-clockwise direction.

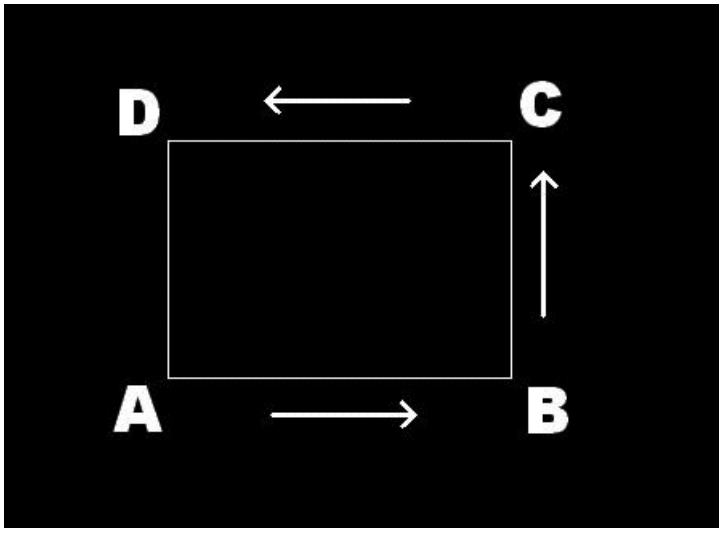
1 From the **Draw** menu, choose **Surface**.

Tip: You can also click the *Surface* tool on the *Draw* toolbar. 

2 Click on the drawing at the starting point of the surface.

3 Moving in a counter-clockwise direction, drag to the next point of the surface and click.

4 Continue to place points for the surface as needed, always moving counter-clockwise. In the following graphic, the starting point is labelled A:



5 To finish and close the surface, right-click and choose **Finish Surface** from the sub-menu. To abort the surface entirely, choose **Abort Surface**. This erases the whole surface from the drawing.

Drawing polygons

Polygons are regular shaped objects comprising multiple vertices. You can draw polygons as 2D objects or specify a height value to draw 3D objects. You can use polygons to create either wireframe or solid objects.

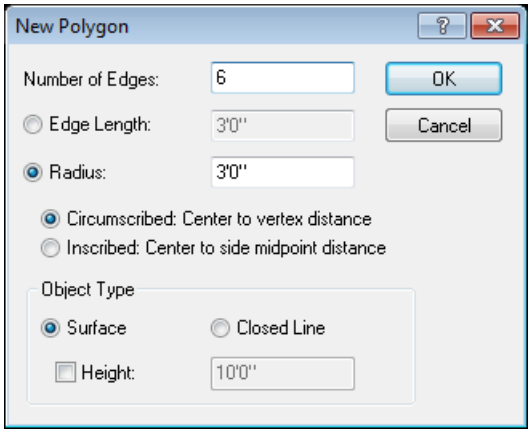
Unlike the Surface tool, with the Polygon tool you can quickly and easily create symmetrical shapes like squares, pentagons, hexagons, hectagons, and so on. The maximum number of edges that a polygon can have is 40.

Notes:

- Unlike most other tools, the polygon tool does not have an interactive mode.
- The size of the polygon can be defined either by specifying the edge length or radius.

1 From the **Draw** menu, choose **Polygon**.

Tip: You can also click the *Polygon* tool on the *Draw* toolbar.



2 In the *Number of Edges* box, type the number of edges or sides of the polygon.

3 Click *Edge Length* if you want to define the size of the polygon by the length of its edges, and then type the value in the box provided.

4 Click *Radius* if you want to define the size of the polygon by the distance between the center and the vertices or the edges' midpoints, and then type the distance in the box provided.

- Click *Circumscribed* if you want to define the size of the polygon by the distance between the center and the vertices. This is equal to the radius of the circumscribed circle.
- Click *Inscribed* if you want to define the size of the polygon by the distance between the center and the edge midpoints. This is equal to the radius of the inscribed circle.

5 Click *Surface* if you want to create the polygon as a surface (i.e., a filled in object to which you can apply a texture, etc.).

6 Click *Closed Line* if you want to create the polygon as a closed line (i.e., a polygon comprised of an outline that is open in the middle).

7 Click *Height* if you want to extrude the polygon into a 3D object, and then type the height value in the box provided.

8 Click *OK* to place the polygon in your drawing.

Drawing walls

Walls are 2D objects that describe flat rectangular planes representing walls in your drawing. Walls can have different hatching styles for easy identification in the plot.

There are two ways to draw a wall:

- non-interactive mode in which you specify the width and height and place the wall in the drawing.
- interactive mode where you draw the wall freehand.

To draw a wall in non-interactive mode

1 From the **Draw** menu, choose **Wall**.

Tip: You can also click the *Wall* tool on the *Draw* toolbar. 

Result: The *New Wall* window will appear.

2 In the *Width* box, enter the width for the wall. The default value is 12’0”.

3 In the *Height* box, enter the height of the wall. The default value is 8’0”.

4 In the Thickness box, enter the thickness of the wall. The default value is 0'6"

5 Click *OK*.

Result: The wall attaches to the cursor.

6 Move the cursor to the desired position for the wall, and then click to place it in the drawing.

To draw a wall in interactive mode

Tip: To turn interactive mode on, click the Interactive Mode button on the toolbar. 

1 From the **Draw** menu, choose **Wall**.

2 Click to place the start point of the wall.

3 Click to place the end point of the wall.

4 In the dialog box that opens, enter the missing dimension of the wall.

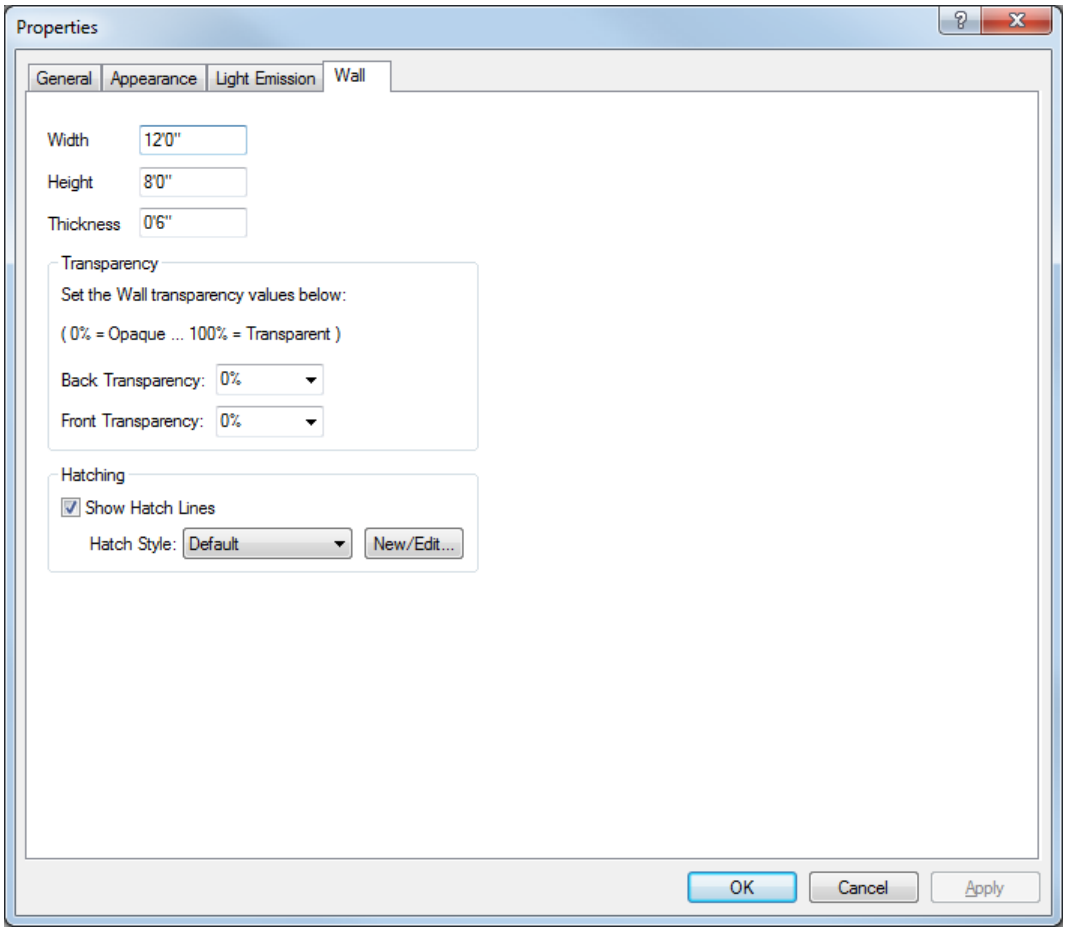
5 Click *OK*.

To define the transparency of a wall

1 Right click a wall, and select **Object Properties**.

Result: The *Properties* window will appear.

2 In the *Properties* window, click the *Wall* tab.



Note: 0% = Opaque, 100% = Transparent.

3 To change how the back of the wall will look, set the *Back Transparency* field to **the desired transparency percentage**.

4 To change how the front of the wall will look, set the *Front Transparency* field to **the desired transparency percentage.**

5 Click *Apply* to enable the new wall transparency settings.

6 Click *OK*.

To define hatching styles

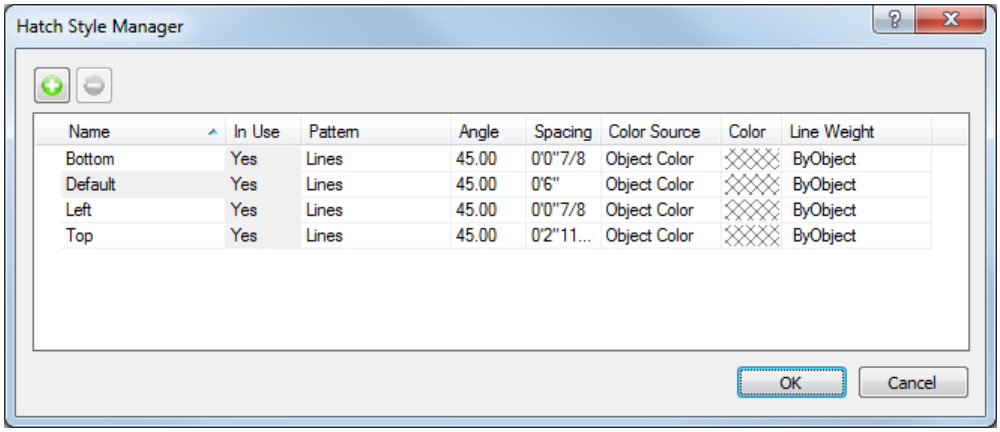
1 Right click a wall, and select **Object Properties**.

Result: The *Properties* window will appear.

2 In the *Properties* window, click the *Wall* tab.

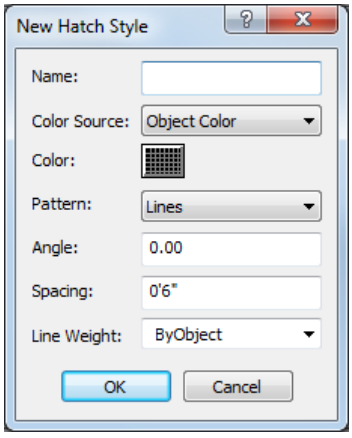
3 To make a new hatch style or edit an existing style, click *New/Edit...* .

Result: The *Hatch Style Manager* window will appear. All existing hatch styles are displayed and can be edited here.



4 To create a new hatch style, click the *New Hatch Style* button .

Result: *The New Hatch Style* window will appear.



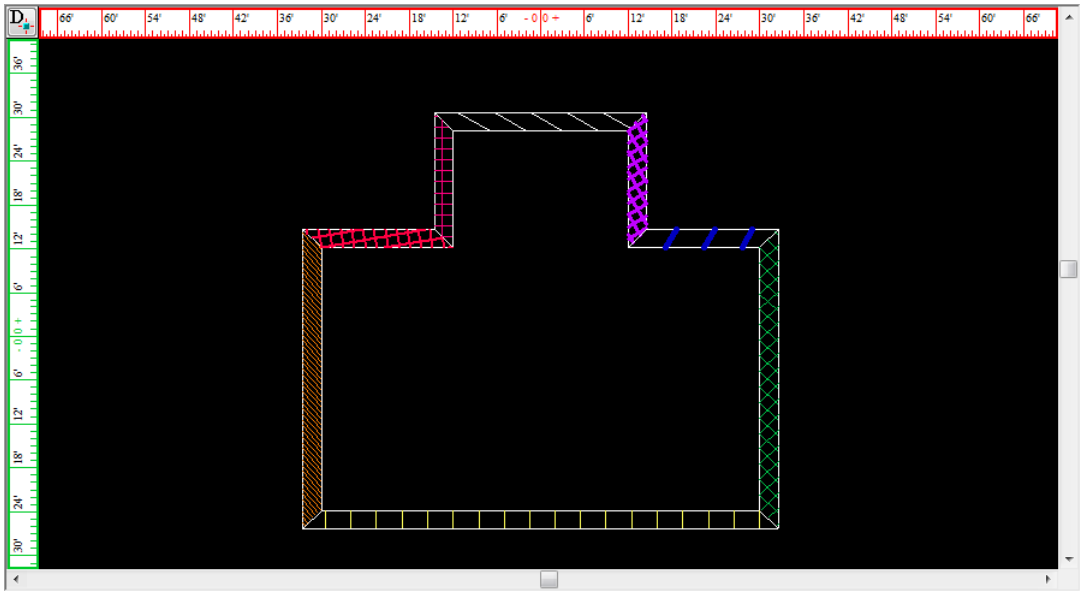
5 In the *New Hatch Style* window, enter the details of the new hatch style.

6 Click *OK*.

7 Click *Apply* to enable the new styles.

8 Click *OK*.

Result: The wall will display the selected hatch style.



Drawing risers

Risers are solid 3D rectangular objects. You can use risers for creating platforms, square columns, or any other box-shaped objects.

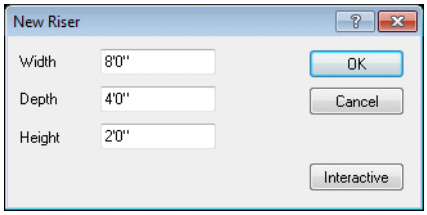
There are two ways in which you can draw risers:

- non-interactive mode in which you type the exact width, depth, and height of the riser
- interactive mode in which you click and drag to set the dimensions of the riser

To draw a riser in non-interactive mode

1 From the **Draw** menu, choose **Riser**.

Tip: You can also click the *Riser* tool on the *Draw* toolbar.



2 In the *Width* box, type the width for the riser. The default value is 8’0”

3 In the *Depth* box, type the depth for the riser. The default value is 4’0”.

4 In the *Height* box, type the height for the riser. The default value is 2’0”.

5 Click *OK*.

Result: The riser is attached to the cursor.

6 Drag the new riser to its position, and then click to place it in the drawing.

To draw a riser in interactive mode

Note: To turn interactive mode on, click the Interactive Mode button on the toolbar. 

1 From the **Draw** menu, choose **Riser**.

2 Click the starting point of the riser on the drawing. The insertion point is the lower left corner of the riser.

3 Drag in any direction to stretch out the riser's shape. Click to place the upper right corner of the riser.

4 In the dialog box that opens, type the missing dimension of the riser.

5 Click *OK*.

Note: You can now snap to all eight points on a riser: top corners and bottom corners.

Drawing cylinders

Cylinders are solid 3D objects. Cylinders can be used for creating platforms, columns, or any other cylinder-shaped objects.

There are two ways in which you can draw cylinders:

- non-interactive mode in which you type the exact height and radius of the cylinder
- interactive mode in which you click and drag to set the dimensions of the cylinder

To draw a cylinder

1 From the **Draw** menu, choose **Cylinder**.

Tip: You can also click the *Cylinder* tool on the *Draw* toolbar. 

2 In the *Height* box, type the height for the cylinder. The default value is 8'0".

3 In the *Horizontal Radius* box, type the horizontal radius for the cylinder. The default value is 4'0".

4 In the *Vertical Radius* box, type the vertical radius for the cylinder. The default value is 4'0".

5 To ensure that the cylinder remains proportionately the same when resized, leave the *Lock ratio* check box checked. If you deselect this check box, then you can manually resize the cylinder in any direction, regardless of its original measurements.

6 To have the cylinder appear smooth in the Shaded view, leave the *Smooth Shading* option enabled; to see a faceted cylinder in the Shaded view, disable this option. The default for this option is ON.

7 To change the default number of cylinder segments, deselect the check mark beside *Use Document Defaults* and type the new number of segments. Note that the higher the number of segments, the better the cylinder will appear in Shaded view, but the more performance will degrade.

8 Click *OK.*

Result: The cylinder is attached to the cursor.

9 Drag the new cylinder to its position, and then click to place the object in the drawing.

To draw a cylinder in interactive mode

Note: To turn interactive mode on, click the Interactive Mode button on the toolbar. 

1 From the **Draw** menu, choose **Cylinder**.

2 Click to place the center of the bottom surface of the cylinder.

3 Click to set the radius for the cylinder.

4 In the dialog box that opens, type the desired height, horizontal radius and vertical radius for the cylinder.

5 To ensure that the cylinder remains proportionately the same when resized, leave the *Lock ratio* check box checked. If you deselect this check box, then you can manually resize the cylinder in any direction, regardless of its original measurements.

6 To have the cylinder appear smooth in the Shaded view, leave the *Smooth Shading* option enabled; to see a faceted cylinder in the Shaded view, disable this option. The default for this option is ON.

7 To change the default number of cylinder segments, deselect the check mark beside *Use Document Defaults* and type the new number of segments. Note that the higher the number of segments, the better the cylinder will appear in Shaded view, but the more performance will degrade.

8 Click *OK*.

Drawing spheres

Spheres are solid 3D circular or oval objects.

There are two ways in which you can draw spheres:

- non-interactive mode in which you type the exact horizontal and vertical radius of the sphere
- interactive mode in which you click and drag to set the dimensions of the sphere

To draw a sphere in non-interactive mode

1 From the **Draw** menu, choose **Sphere**.

Tip: You can also click the *Sphere* tool on the *Draw* toolbar. 

2 In the *Horizontal Radius* box, type the horizontal radius for the sphere.

3 In the *Vertical Radius* box, type the vertical radius for the sphere.

4 To ensure that the sphere remains proportionately the same when resized, leave the *Lock ratio* check box checked. If you deselect this check box, then you can manually resize the sphere in any direction, regardless of its original measurements.

5 To have the sphere appear smooth in the Shaded view, leave the *Smooth Shading* option enabled; to see a faceted sphere in the Shaded view, disable this option. The default for this option is ON.

6 To change the default number of sphere segments, deselect the check mark beside *Use Document Defaults* and type the new number of segments. Note that the higher the number of segments, the better the sphere will appear in Shaded view, but the more performance will degrade.

7 Click *OK*.

Result: The sphere is attached to the cursor.

8 Click to place the sphere in the drawing.

To draw a sphere in interactive mode

1 From the **Draw** menu, choose **Sphere**.

Tip: You can also click the *Sphere* tool on the *Draw* toolbar. 

2 In the *New Sphere* window, click *Interactive*.

3 Accept the default values, and then click and drag out the shape of the sphere in your drawing. When you have the desired size, click again.

Result: The *New Sphere* window appears again.

4 In the dialog box that opens, type the horizontal and vertical radius of the new sphere.

5 To ensure that the sphere remains proportionately the same when resized, leave the *Lock ratio* check box checked. If you deselect this check box, then you can manually resize the sphere in any direction, regardless of its original measurements.

6 To have the sphere appear smooth in the Shaded view, leave the *Smooth Shading* option enabled; to see a faceted sphere in the Shaded view, disable this option. The default for this option is ON.

7 To change the default number of sphere segments, deselect the check mark beside *Use Document Defaults* and type the new number of segments. Note that the higher the number of segments, the better the sphere will appear in Shaded view, but the more performance will degrade.

8 Click *OK*.

9 Click in the drawing to finish placing the new sphere.

Drawing cones

Cones are solid 3D objects.

There are two ways in which you can draw cones:

- non-interactive mode in which you type the exact horizontal and vertical radius of the cone
- interactive mode in which you click and drag to set the dimensions of the cone

To draw a cone in non-interactive mode

1 From the **Draw** menu, choose **Cone**.

Tip: You can also click the *Cone* tool on the *Draw* toolbar. 

2 In the *Height* box, type the height of the cone.

3 In the *Horizontal Radius* box, type the horizontal radius for the cone.

4 In the *Vertical Radius* box, type the vertical radius for the cone.

5 To ensure that the cone remains proportionately the same when resized, leave the *Lock ratio* check box checked. If you deselect this check box, then you can manually resize the cone in any direction, regardless of its original measurements.

6 To have the cone appear smooth in the Shaded view, leave the *Smooth Shading* option enabled; to see a faceted cone in the Shaded view, disable this option. The default for this option is ON.

7 To change the default number of cone segments, deselect the check mark beside *Use Document Defaults* and type the new number of segments. Note that the higher

the number of segments, the better the cone will appear in Shaded view, but the more performance will degrade.

8 Click *OK*.

Result: The cone is attached to the cursor.

9 Click to place the cone in the drawing.

To draw a cone in interactive mode

1 From the **Draw** menu, choose **Cone**.

Tip: You can also click the *Cone* tool on the *Draw* toolbar. 

2 In the *New Cone* window, click *Interactive*.

3 Accept the default values, and then click and drag out the shape of the cone in your drawing. When you have the desired size, click again.

Result: The *New Cone* window appears again.

4 In the dialog box that opens, type the height and the horizontal and vertical radius of the new cone.

5 To ensure that the cone remains proportionately the same when resized, leave the *Lock ratio* check box checked. If you deselect this check box, then you can manually resize the cone in any direction, regardless of its original measurements.

6 To have the cone appear smooth in the Shaded view, leave the *Smooth Shading* option enabled; to see a faceted cone in the Shaded view, disable this option. The default for this option is ON.

7 To change the default number of cone segments, deselect the check mark beside *Use Document Defaults* and type the new number of segments. Note that the higher the number of segments, the better the cone will appear in Shaded view, but the more performance will degrade.

8 Click *OK*.

9 Click in the drawing to finish placing the new cone.

Drawing pipes

For details, refer to “Drawing pipes” on page 259.

Drawing masking

Masking, specifically borders and legs, can be automatically attached to pipes using the border and legs tool.

To draw borders and legs

1 Select the pipes for which you want to attach a border and legs.

2 From the **Tools** menu, choose **Border and Legs**.

3 Select the *Border* check box to add a border to the selected pipes, and then specify the height of the border.

Note: This is the distance from the pipe to the bottom of the border.

4 Select the *Legs* check box to add legs to the selected pipes, and then specify the width and bottom elevation of the legs.

Note: The width is the distance from the extremity of the pipe to the inside edge of the leg. The bottom elevation is the distance from the floor (z = 0) to the bottom of the legs.

5 Select the *Group to pipe* check box to group the border and legs to the pipe. When this option is selected all the components (border, legs, and pipe) are selected as one.

Note: If, at a later time, you want to edit a border or leg individually, you must ungroup the objects.

6 Click *OK*.

Result: Three surfaces are drawn and attached to the selected pipe.

Drawing focus objects

For details on drawing focus objects, such as focus positions, focus lines and focus arcs, refer to “Focus Objects” on page 283.

Drawing screens

You can draw a screen and then attach an image or video source or subsource to it.

To draw a screen

1 From the **Draw** menu, click **Screen**.

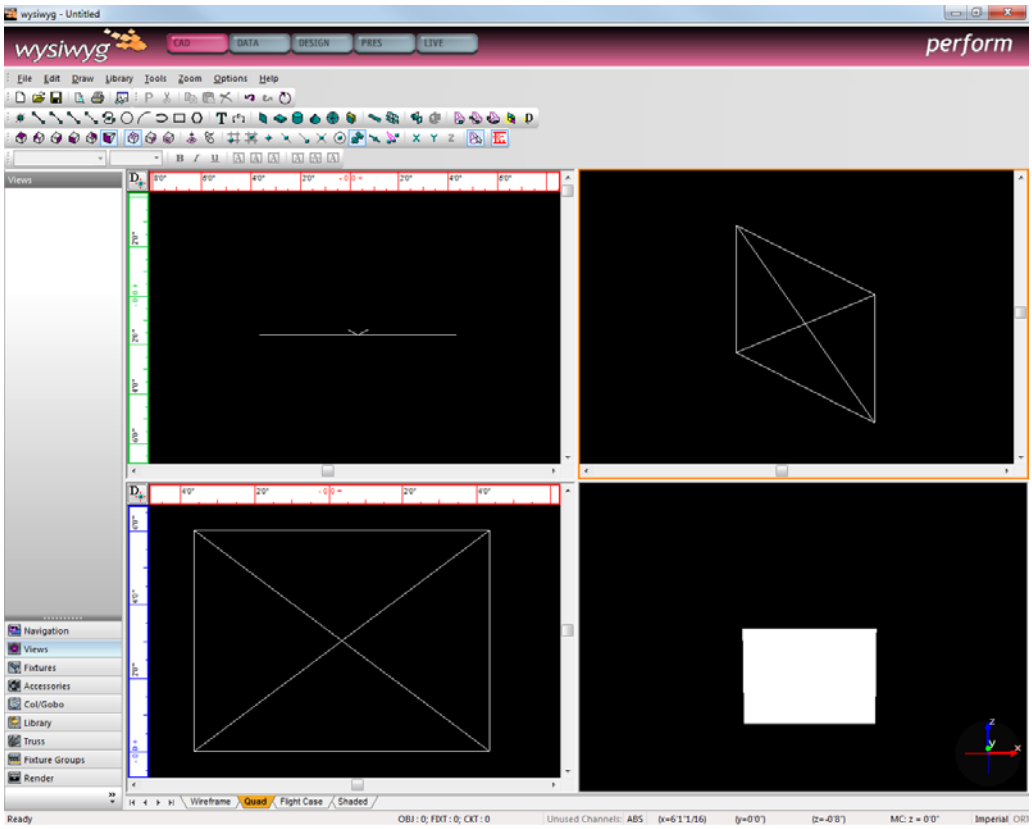
2 In the appropriate boxes, type the width and height of the screen.

3 To configure the image that will appear on the screen, click the appropriate option button:

- To leave the screen blank, click the *Blank Screen* option button.
- To show a static image on the screen, follow the procedures in the Image Manager section. For details, see “Image Manager” on page 343.
- To attach a video source to the screen, click the *Video Source* option button, and then choose the video source or video subsource from the drop-down box (or click *Create New* to configure a new video source with the Video Manager). For details on creating a new video source, see “To configure a new video source for streaming video” on page 341. For details on splitting video into subsources, see “To create a single video subsource” on page 338.

4 Click *OK*.

Result: The screen is placed into the drawing.



Drawing cameras

You can add cameras to the drawing to provide additional view points in shaded views and renderings.

Note: For tips on moving your cameras, see “Camera controls” on page 78.

To insert a camera

Note: The following procedure outlines how to insert a camera in a wireframe view. For details on creating a camera directly from the Shaded view, see “To create a camera from shaded view” on page 80.

1 From the **Draw** menu, choose **Camera**.

2 Type a name for the camera and a value in degrees for the field angle. The default value for the field angle is 53.13.

3 From the *Layer* drop-down box, select the layer of your file on which you want to draw the camera.

Note: If you do not see all layers in the *Layer* drop-down box, click *Show non-editable layers*.

4 Click on the drawing to place the camera.

5 Drag the cursor to aim the camera, and then click to place the target.

Result: The camera is inserted into your drawing and available in the camera selection list for shaded views and renderings.

Tip: Use the missing coordinate tool while placing the camera and target.

Note: The target of your camera is easily identifiable in Wireframe views as defined by the marker points of the camera drawing. Camera drawings are invisible in shaded views because you are looking *through* the camera. For this reason, in

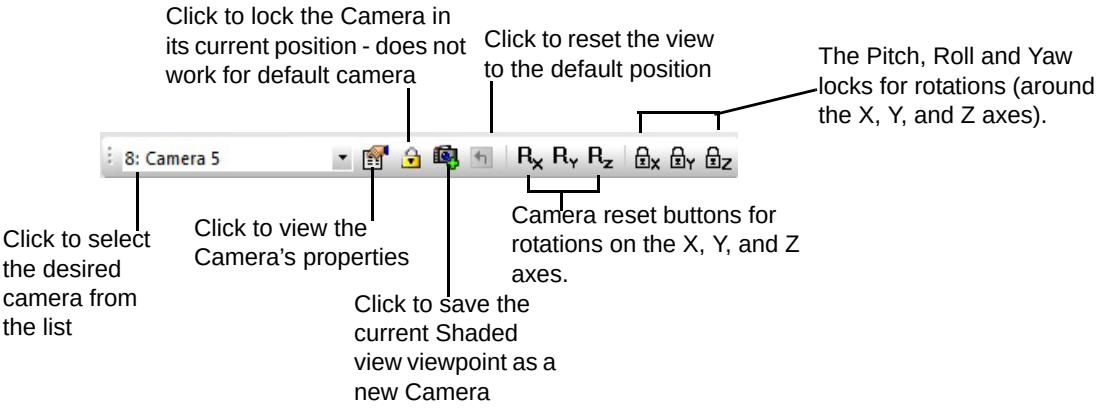
this view you can choose to display the target. For more information on displaying the target, refer to “Modifying shaded views” on page 81 or see “Using the Camera toolbar” on page 165.

Using the Camera toolbar

After you have drawn a camera, you can use the Camera toolbar in Shaded views to view your drawing through the camera of your choice.

You can also use this toolbar to quickly access the camera Properties page and change properties, such as the camera name and field angle, and to lock the camera in place so that you cannot accidentally move it. For details, see the procedures on the next page.

Finally, the Camera toolbar also enables you to quickly insert a camera directly from the shaded view, based on the current viewpoint. For details, see “To create a camera from shaded view” on page 80.



To switch to a camera using the Camera toolbar

1 In CAD, Design, or Live mode, click the Quad or Shaded tab.

2 Right-click in the toolbar area and select **Camera**.

Result: The Camera toolbar appears.

3 From the *Current Camera* drop-down list, select the camera through which you want to view your model.

Note: To switch back to the default view, select None, and then click the far-right button (*Reset*) to reset the view.

Result: The viewpoint switches to reflect that of the currently selected camera.

To use Camera hotkeys

To help you quickly switch from one Camera to another, Number/Numpad keys 1 through 9 are now automatically assigned to new Cameras as you create them.

Hotkeys are assigned in ascending order (starting with 1 for the first Camera you create, 2 for the next one, and so on, all the way to 9).

1 On the number pad of your keyboard, ensure that the Num Lock is activated.

2 In WYSIWG, click to view your drawing in a Shaded view.

3 Either on the Number Pad of your keyboard, or using the regular number buttons, press the numbers associated with the Cameras in your drawing to switch quickly from one Camera to another.

Tip: Press the 0 key to quickly switch to the World camera.

To change or delete the Camera hotkey assignment

To change or delete the hotkey assignment for a Camera, complete the following steps.

1 Select the Camera, right-click, and select **Properties**.

2 Click the Camera tab.

3 From the *Assigned Hotkey* drop-down box, select the new hotkey; to remove a hotkey assignment, simply choose None from the same drop-down.

4 Click *OK*.

To view or edit Camera Properties

1 In CAD, Design, or Live mode, click the Quad or Shaded tab.

2 Right-click in the toolbar area and select **Camera**.

Result: The Camera toolbar appears.

3 From the *Current Camera* drop-down list, select the camera whose Properties you want to view.

4 On the Camera toolbar, click the *Camera Properties* button.

5 In the window that appears, you can change any of the following properties:

- *Name*: You can edit the name that appears in the Camera toolbar for the currently selected Camera.
- *Field Angle*: You can change the field angle of the currently selected Camera, measured in degrees.
- *Assigned Hotkey*: Click to assign a new numbered hotkey for this Camera. This is the number that you can press to instantly switch from one Camera to the next.
- *Lock camera in Shaded View*: Click this check box to lock the selected camera in its current position in all shaded views. To unlock it, simply select the camera from the Camera toolbar, and then press the *Camera Lock* button.

To lock a Camera

Follow the steps below to lock a Camera in its current position in the shaded view. This feature helps prevent you from accidentally moving or zooming a camera while adjust the viewpoint in the shaded view.

1 In CAD, Design, or Live mode, click the Quad or Shaded tab.

2 Right-click in the toolbar area and select **Camera**.

Result: The Camera toolbar appears.

3 From the *Current Camera* drop-down list, select the camera that you want to lock in position.

4 On the Camera toolbar, click the *Camera Lock* icon. 

Result: The Camera is now locked in position in the shaded view. Whenever you select this camera and attempt to adjust the viewpoint in the shaded view, a lock symbol appears in the middle of the shaded view. To unlock the Camera, simply select it from the Camera toolbar, and press the *Camera Lock* icon again.

To use the Pitch, Roll and Yaw locks for rotations

Use the Lock-X, Lock-Y, and Lock-Z buttons on the Camera toolbar to prevent in-place rotations along the specified axis:

- Lock-X, or Rotation around the X Axis = PITCH [tilting forward and backward]
- Lock-Y, or Rotation around the Y Axis = ROLL [tilting side to side]
- Lock-Z, or Rotation around the Z Axis = YAW [also called Heading or Direction]

To see the effect of these locks in the Shaded view, you must press ALT while dragging the view; these locks have no effect if you do not press ALT.

Note: Press the reset buttons (Rx, Ry, Rz) to re-orient the camera to the world's axes.

Drawing Camera paths

Camera paths allow you to define a path along which the camera will travel during playback in the Shaded View, either in Design or Live Modes. You draw Camera paths in CAD Mode just as you would draw a spline. Each point along the path is called a “Node”.

Note: You can keep the camera path, nodes and targets visible during file editing, and once you’re done, go into the Properties of these objects and toggle their visibility off in Shaded View.

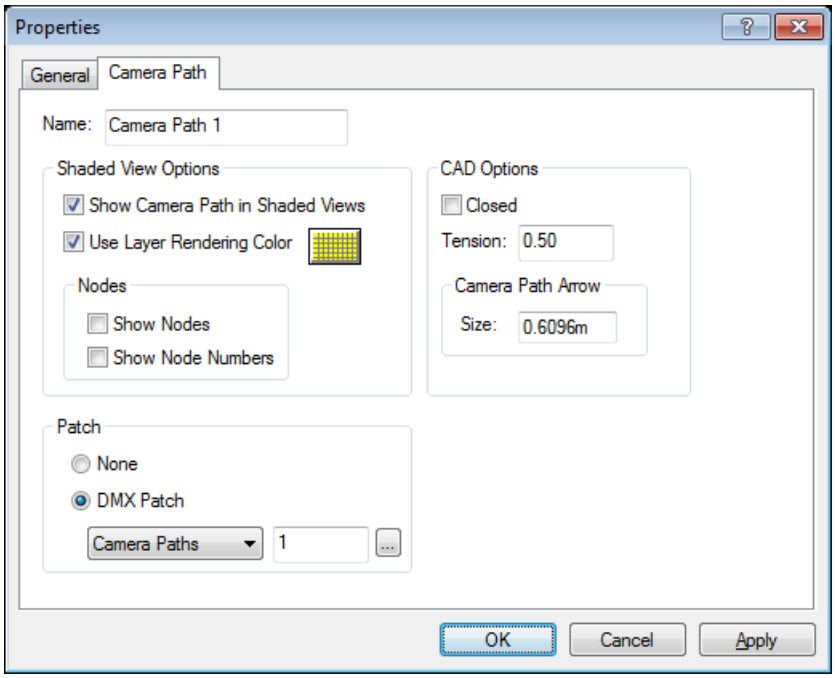
To draw a Camera path

1. In Wireframe view, from the **Draw** menu, click **Camera** > **Camera Path**.
2. Click in your plot where you want to start drawing the Camera path.
3. Continue clicking in your drawing to create your path. When you are finished, right-click and select one of the options on the pop-up menu:
   - **Finish Camera Path**: This option finishes the path at the last point, and places it in your plot.
   - **Close Camera Path**: This option closes the path from one end to the next, and places the path in your plot.
   - **Abort Camera Path**: This option cancels the drawing of the Camera path altogether.
   - **Remove Last Step**: This option removes the last point of the path.

   Note: Camera paths are controlled from Design and/or Live modes, using the Camera Path Editor and the Timeline tool. For details, see “Using the Camera path editor” on page 493.

Camera Path properties

Camera Paths have unique properties, as shown below:



- *Name*: Camera Paths can have unique names
- *Closed*: If checked, this option enables the Camera Path to play as a continuous loop with no gaps.
- *Tension*: A value between 0 and 1, where 0 is no tension and 1 is maximum tension; this option enables the Camera Path to allow curving between nodes along the path. If the Tension is set to 0, the nodes will be connected with straight lines.
- *Show Camera Path in Shaded Views*: This option shows or hides the Camera Path in Shaded Views while playing the path.
- *Use Layer Rendering Color*: This option controls the color of the Camera Path and its Layer behaviour.
- *Show Nodes*: This option shows or hides the Nodes along the Camera Path.
- *Show Node Numbers*: This option shows or hides the Node numbers along the Camera Path.
- *Camera Path Arrow Size*: This option allows you to control the size of the direction arrows on the Camera Path.
- *Patch - None*: Select this option if the Camera Path is not patched.
- *DMX Patch*: Select this option if the Camera Path is patched, and then enter the patch properties.

Drawing Camera targets

After you draw a Camera path, if you wish, you can also insert a “Camera target” at points of interest where you would like the camera to focus while travelling along a path.

Note: You can keep the camera path, nodes and targets visible during file editing, and once you’re done, go into the Properties of these objects and toggle their visibility off in Shaded View.

To draw a Camera target

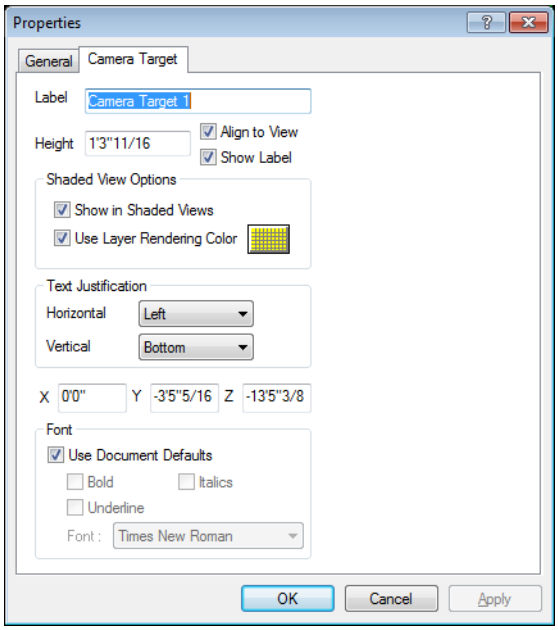
1 In Wireframe view, from the **Draw** menu, click **Camera** > **Camera Target**.

2 Type a descriptive name for the camera target.

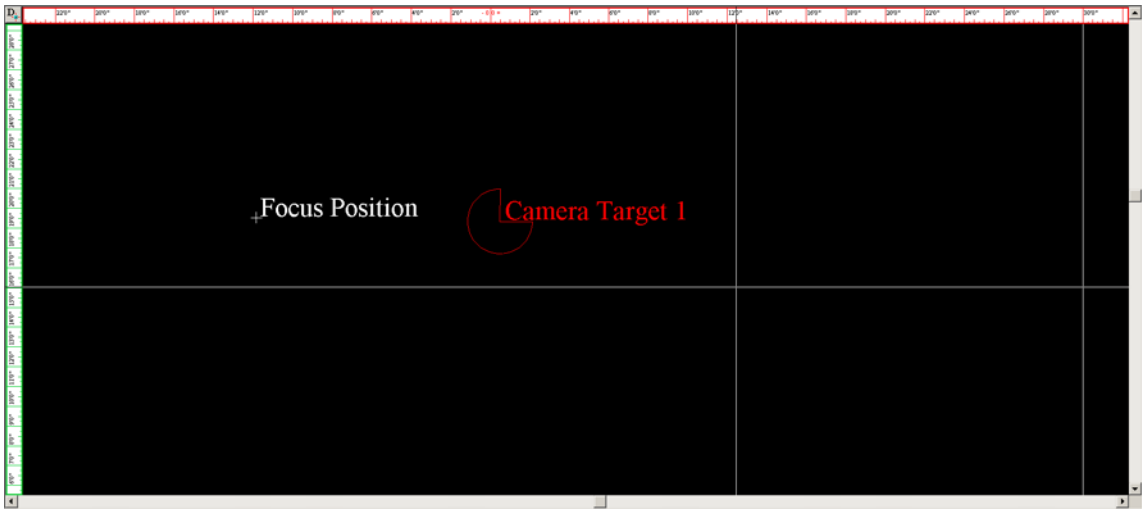
3 Click *OK* and then click to place the target along the camera path.

Camera target properties

Camera targets also have unique properties, as seen below:



- *Label*: Camera targets can have unique labels.
- *Height*: This controls the height of the text associated with the Camera Target.
- *Align to View*: This option aligns the target to the specific view that it is in (i.e., Side, Front, Back, Iso)
- *Show Label*: This option shows or hides the Camera Target label text.
- *Show in Shaded Views*: This option shows or hides the Camera Target in Shaded Views while playing the path.
- *Use Layer Rendering Color*: This controls the color of the Camera target and its Layer behaviour.
- *Text Justification*: This option controls the behavior of the text associated with the Camera Target.
- *X, Y, and Z Coordinates*: This option allows you to control the position in 3D space of the Camera Target.
- *Font*: This option allows you control of the font face of the Camera Target, and uses Document Defaults by default.



Drawing a compass

You can add a compass to a drawing to define the direction that is north. This definition is important to set the position of the sun and/or moon for renderings and Shaded views that use environmental options.

Note: There can be only one compass in a file.

To draw a compass

1 From the **Tools** menu, choose **Define North**.

2 Click to place the starting point of the compass.

3 Click to place the end point of the compass. The end point must be directly north of the start point.

Result: A compass is drawn, centered on the start point and pointing towards the end point.

Notes:

- To remove the compass from the drawing, choose **Clear Direction** from the **Tools** menu.
- To redefine north, repeat steps 1-3. You do not need to remove the compass from the drawing when redefining north as there can be only one compass in a file.

Drawing motion axes

For every motion axis that you draw in CAD mode, you can attach one or more objects to it, and then define the object's position along the axis. You can attach objects such as risers, library objects, or custom objects that you have drawn.

To be able to move an object in different planes, you can attach multiple motion axes to one another. Simply attach the object to one of the axes, and then control the object's movement along any of the axes by selecting it with the Moving Scenery Designer tool. This feature is useful, for example, if you have a set piece that needs to be able to move in both a vertical and horizontal motion.

Notes:

- When attaching axes to one another, you cannot create a circular linking pattern. For example, you can link Axis A to Axis B, and then link Axis B to Axis C, but you cannot then link Axis C back to Axis A because this creates a circular link.
- You cannot attach fixtures to motion axes; however, you can attach pipes that hold fixtures to axes.
- Currently, you cannot attach focus positions or groups containing focus positions to motion axes. However, in future releases you will be able to attach either of these items to motion axes.

You can draw axes either diagonally or straight along any direction of movement—X, Y, or Z—choosing between the following two main types of axes:

- **Linear:** Choose this type of axis to have an object move in a straight line along the path that you specify (or in a series of straight segments all joined together at different vertices to create the path). You can create a linear axis in any length or shape that you require, as long as it comprises straight line segments; you cannot draw a curved axis. For example, you could have a person walk across a stage in a zig zag path, if desired.

  When you draw this type of axis, and then attach an object to it, the distance from the axis to the object does not affect the object's movement; the object will always follow the path of the axis regardless of where you have placed the axis or object. By default, all linear axes have an arrow on one end, signalling the direction in which the object will move. You can adjust the size of the arrow to suit your needs by opening the Properties page for the axis.
- **Rotation:** Choose this type of axis if you want to have an object rotate in a circular motion. When you click to place this type of axis in your drawing, a circle with a center dot appears. The circle acts as a visual cue, telling you the direction in which the object will rotate; the center dot is the actual axis around which the object rotates.

  The orientation of the object's rotation changes based on the view in which you draw the axis (plan, side, or front/back). When placing the object to be rotated in your drawing, note that its position relative to the center dot of the rotation axis is important. The distance from the center dot to the object represents the size of the circle in which it will rotate, with the dot marking the center of the circle. To have the object rotate in a very tight circle (for example, a dancer doing a pirouette, or a spinning object), place the object directly on the center dot.

  By default, all rotation axes have an arrow on one end, signalling the direction in which the object will rotate. While you cannot adjust the size of the arrow head directly, you can click and drag the circle to the desired size, which adjusts the arrow head proportionately. To change the direction of the object's rotation, switch views, and then drag the center point to the new orientation.

To draw a linear motion axis

1. In Wireframe view, select the view in which you want to draw the axis (plan, front, back, side, or isometric)
2. Click **Draw** > **Axis** > **Linear Axis**.
3. In the window that appears, type a name for the axis, and then click *OK*.
4. In your drawing, click in the position where you want to start the axis, and then move your cursor to the next point of the axis and click. To create an axis with multiple segments, continue clicking at each point (vertex) of the axis.
5. When you are finished creating the axis, right-click and choose **Finish Axis**.

To draw a rotation motion axis

Note: Unlike linear axes, rotation axes cannot have multiple vertices. Instead, when you draw one of these axes, the object rotates around the center point.

1 In Wireframe view, select the view in which you want to draw the axis (plan, front, back, side, or isometric).

2 Click **Draw** > **Axis** > **Rotation Axis**.

3 In the window that appears, type a name for the axis, and then click *OK*.

4 In your drawing, click in the position where you want the axis to appear.

Note: The circle that appears when you draw a rotation axis is only a visual cue to show you the direction in which the object rotates—it does not affect the size of path on which the object rotates. The circle is given a default size, but you can change it to suit your needs.

To patch a motion axis to a DMX universe

To move the object with a DMX control, you first have to attach the motion axis to a named DMX patch universe that you have created in Data mode.

Note: For information on creating a patch universe, see “To create a new patch universe” on page 367.

1 Click to select the axis that you want to attach to the patch universe.

2 Right-click and select **Properties**.

3 Click the Axis tab.

4 From the *Patch Type* drop-down box, select *DMX Patch*.

5 From the *Universe* drop-down box, select the DMX patch universe to which you want to attach the axis.

6 In the *Address* box, type the starting DMX address.

7 Click *OK*.

To patch a motion axis to a motion universe

To move the object with a motion control system, you first have to attach the motion axis to a named motion control patch universe that you have created in Data mode.

Note: For information on creating a patch universe, see “To create a new patch universe” on page 367.

1 Click to select the axis that you want to attach to the patch universe.

2 Right-click and select **Properties**.

3 Click the Axis tab.

4 From the *Patch Type* drop-down box, select *Motion Patch*.

5 From the *Universe* drop-down box, select the motion control patch universe to which you want to attach the axis.

6 From the *ID* drop-down list, select the ID on the motion control system with which you want to associate this axis.

Note: The motion control system must be connected to your computer for the ID to appear in this drop-down list. If the system is not currently connected, you can also type the ID directly into this drop-down list box.

7 Click *OK*.

To attach an object to a motion axis

Notes:

- You can attach one or more objects to each axis; however, each object can be attached to only one axis at a time.

- When you attach an object to a linear axis, the distance from the axis to the object does not affect the object's movement; the object will always follow the path of the axis regardless of where you have placed the axis or object.
- When placing the object to be rotated in your drawing, note that its position relative to the rotational axis is important. The distance from the central point of the axis to the object represents the size of the circle in which it will rotate. To have the object rotate in a very tight circle (for example, a dancer doing a pirouette, or a spinning object), place the object directly on the center point of the axis.

1. Place the object that you want to attach to the axis at the appropriate position in your drawing, either by adding a new object, or moving an existing object to this location.
2. Select the object, right-click, and then choose **Properties**.
3. On the *General* tab, from the *Attach to Axis* drop-down box, select the axis to which you want to attach this object.
4. Click *OK*.

To add a vertex to a linear motion axis

To change the path of an existing linear axis, you can add one or more vertices, and then drag these vertices to the new position.

1. In your drawing, click on the point of the axis at which you want to add a vertex.
2. Right-click and select **Add Vertex**.

   Result: The vertex is added at the point where you clicked.

   Tip: To change the path of the axis, click to select the axis and view all the vertices. Click the desired vertex, drag it to the new position, and then release the mouse. Any objects that are attached to the axis will now follow the new path.

Motion Frames

The Frame Motion System delivers more functionality and flexibility than its predecessor, the Axis system. A single motion frame can simulate motion for up to six Degrees of Freedom (DoF) for each frame.

Much like the motion axis, you can attach one or more objects to a single motion frame, and then define the motion ranges of each DoF.

Notes:

- When attaching one frame to another, you cannot create a circular linking pattern.
- You cannot attach fixtures to frames; however, you can attach pipes that hold fixtures to axes.

To draw a motion frame

1. In a Wireframe view, select the view in which you want to draw the frame.
2. Click **Draw** > **Frame.**
3. In the window that appears, type the name of the frame, and then click *OK*.
4. In your drawing, click in the position where you want the frame to appear.

To adjust the motion range

1 In a Wireframe view, select the frame that you want to adjust.

2 Right-click and select **Properties**.

3 Select the Motion Ranges tab.

4 To restrict the motion frame, click to enable/disable any of the six Degrees of Freedom (DoF) shown, and then adjust the movement ranges of each DoF by modifying the Minimum and Maximum values for each corresponding axis.

5 To enforce these motion ranges while controlling objects in LIVE mode, click to enable the *Use Motion Ranges in Live Mode* checkbox.

6 Click *OK*.

To patch a motion frame to a DMX universe

To move the object with a DMX control, you have to attach the motion frame to a named DMX patch universe that you have created in Data mode. Patching a single motion frame creates 12 DMX channels: X, X Fine, Y, Y Fine, Z, Z Fine, Rx, Rx Fine, Ry Fine, Rz Fine.

Note: For information on creating a patch universe, see “To create a new patch universe” on page 367.

1 Click to select the axis that you want to attach to the patch universe.

2 Right-click and select **Properties**.

3 Click the Axis tab.

4 From the *Patch Type* drop-down box, select *DMX Patch.*

5 From the *Universe* drop-down box, select the DMX patch universe to which you want to attach the axis.

6 In the Address box, type the starting DMX address.

7 Click *OK*.

To patch a motion axis to a motion universe

To move the object with a motion control system, you first have to attach the motion axis to a named motion control patch universe that you have created in Data mode.

Note: For information on creating a patch universe, see “To create a new patch universe” on page 367.

1 Click to select the frame that you want to attach to the patch universe.

2 Right-click and select **Properties**.

3 Click the Axis tab.

4 From the *Patch Type* drop-down box, select *Motion Patch*.

5 From the *Universe* drop-down box, select the motion control patch universe to which you want to attach the axis.

6 From the *ID* drop-down box, select the ID on the motion control system with which you want to associate this axis.

Note: The motion control system must be connected to your computer for the ID to appear in this drop-down list. If the system is not currently connected, you can also type the ID directly into this drop-down box.

7 Click *OK*.

To attach an object to a motion frame

Notes:

- You can attach one or more objects to each frame; however, each object can be attached to only one frame at a time.
- When placing the object to be rotated in your drawing, note that its position relative to the frame is important. The distance from the central point of the frame to the object represents the size of the circle in which it will rotate.
- To have the object rotate in a very tight circle (for example, a dancer doing a pirouette, or a spinning object), place the object directly on the center point of the frame.

1. Place the object that you want to attach to the frame at the appropriate position in your drawing, either by adding a new object, or moving an existing object to this location.
2. Select the object, right-click, and then choose **Properties**.
3. On the General tab, from the *Attach to Axis* drop-down box, select the Frame to which you want to attach this object.
4. Click *OK*.

Viewing moving scenery

After you create the axes/frames and attach the objects to them, you can watch the objects move along the axes/frames by using the Looks feature in Design mode. Just as you can simulate moving lights by cross-fading between one lighting look to the next, you can also simulate moving scenery by creating Looks that contain motion axes/frames and fading from one Look to the next. For more information, see “To view moving scenery” on page 489.

You can also patch the axis in Data mode so you can control the movement of the object with either a DMX console device, such as a Mini Console, or a motion control system. For details on patching motion axes, see “To patch a motion axis to a DMX console device” on page 370 or “To patch a motion axis to a motion control system” on page 371.

Creating complex motion systems

To create more complex motion systems (for example, to have an object revolve around itself or around another object while it is also moving across the stage), you can attach motion frames to axes, or vice versa (you can attach axes to motion frames).

Saving custom library items

Custom library items are made of one or more objects, including custom items that you have drawn, as well as existing library items. You can save a custom library item for a complex object that you have drawn and think you might want to use again.

When you save custom library items, all of the item’s original properties are saved, including its texture and color. In addition, if the item that you are saving is broken down into separate elements, then these elements are also saved with the item and can be viewed on the Appearance tab. For example, if you are saving an object like a riser, each of the four riser elements is saved with the custom library item.

Notes:

- You cannot save fixtures or hanging structures, such as pipes and truss, as components of custom library items.
- You cannot rename the elements of custom library items in the Appearance tab; therefore, it is recommended that you give elements custom names before you

save the item as a custom object. For details, see “To rename object elements” on page 199.

To create a custom library item

1 Select the objects to be saved as a custom item.

2 From the **Library** menu, choose **Create New Library Item**.

Tip: If the menu item is greyed out, one or more of the objects selected cannot be included in the custom library item.

3 Pick a base point for the item. This will become the insertion point for the item.

Result: The Library Item Wizard opens displaying a rotating shaded view of your new item.

4 Type a name for the new library item.

5 To make this item available to other documents, check the appropriate box.

6 Click *Next*.

7 Add or modify the path, if desired.

Note: The path is the location in the library where the item will be stored. An item assigned to the path “Custom/Tables” would be available in the custom section of the items library in a sub-category named Tables.

8 Click *Next*.

9 Type all pertinent information including manufacturer, weight, and any notes.

10 Click *Next*.

11 Type your name.

12 Click *Finish*.

Result: The custom item is placed in the item library as per the path you specified in step 7.

To delete a custom library item

If you do not want to keep a custom library item any longer, you can delete it from the library browser.

Notes:

- You cannot delete a custom library item if it is currently inserted in your plot; instead, you must delete the item from your plot, save and close the file. When you reopen the file you can delete the item from the library browser.
- When you delete a custom library item, you do not delete the shortcut associated with it. You must delete shortcuts manually.

1 In the Library Browser, navigate to the folder where you stored the custom library item.

2 Highlight the library item name, and then right-click and choose **Delete**.

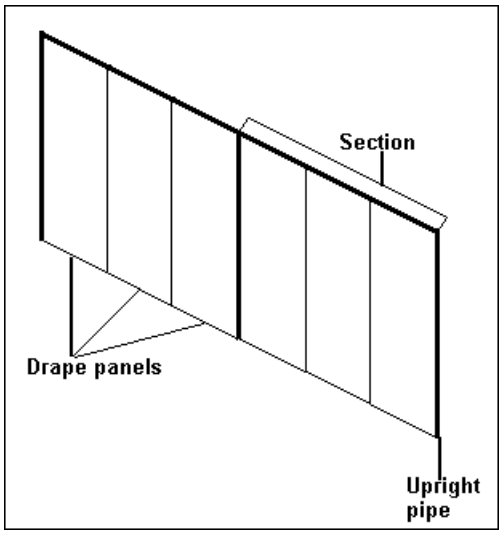
Note: This option is disabled if the item is currently inserted in your plot. You must delete the item from your plot, and then save and close the file. When you reopen the file you can delete the item from the library browser.

3 A message box appears telling you the action is irreversible. Click *Yes* to delete the item.

Drawing pipe and drape

You can add lengths of drapery to your drawing by using the Pipe and Drape Wizard. The Wizard gives you the flexibility of either quickly adding the default length and height of drapery, or specifying custom sizes for your drapery panels. You can also define drapery properties such as color, texture, and material after you have inserted the drapery in your drawing.

There are several components in a drape line, as shown in the following graphic:



- **Drape panels:** These are the individual pieces of fabric that hang from the horizontal pieces of pipe called cross members. You can specify the number of drape panels in each of the pipe and drape drawing methods. The default number of panels is three per section.
- **Section:** A section consists of a series of drape panels hanging from a horizontal pipe called a cross member, which is supported on each end by an upright pipe. You can specify the number of panels per section and, based on the drawing method you choose, either the exact width of the section or the maximum width.
- **Upright pipes:** These are the pipes that stand on either end of each section in your drape line, supporting the cross-member pipes.

The Wizard offers you the following three drawing options:

- **AutoDrape:** This option is similar to the Truss Wizard in that it first enables you to specify simple dimensions. Then, the object of the defined size, in this case a drape line, is attached to your cursor for insertion in the drawing. You can place the drape line wherever you want in your drawing. This option enables you to either draw a new drape line section, or add to an existing drape line.
- **Draw:** This option gives you the most flexibility when drawing your drape line. It enables you to specify the height, maximum width of a section, and the number of drape panels per section. When you click Finish, the Wizard returns you to the drawing where you can click and drag to draw as many drape lines as desired in whatever length required.
- **Specify:** This option enables you to define the height, width, the number of sections in your drape line, and the number of drape panels per section. This option enables you to either draw a new drape line section, or add to an existing drape line.

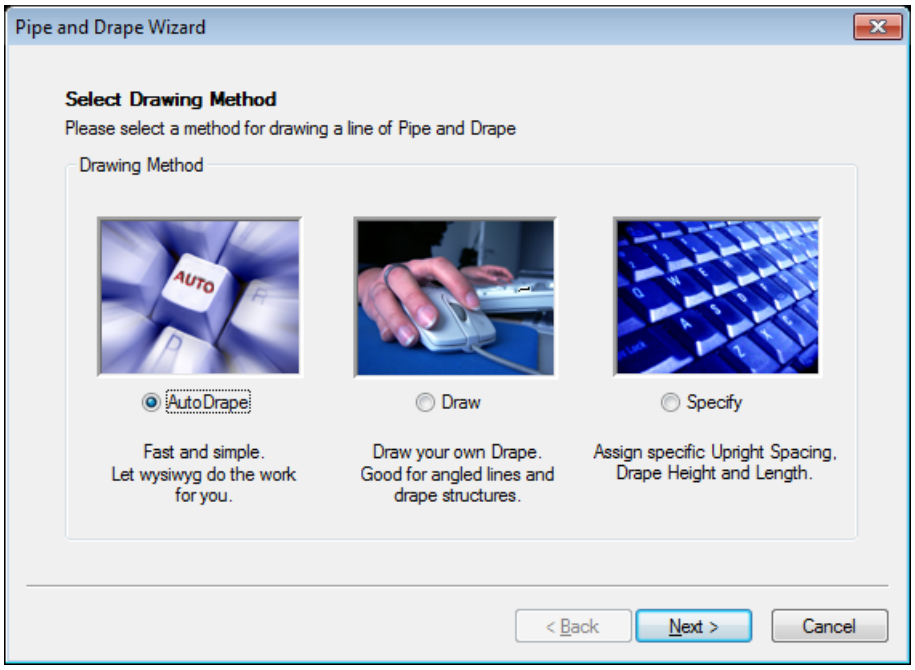
Consult the appropriate procedure below for each of these options.

To draw pipe and drape - AutoDrape method

The AutoDrape method lets you specify the length, height, and number of panels per section in your drape panel; it lets you add one drape section at a time. After you draw the drape panel, you can further customize it by adding color, texture, and material to it.

Note: If you are adding a new drape section to an existing panel in your drawing, click the mid-point snap tool first. This ensures that the new drape panel “snaps” to the existing one in the correct position.

1 From the **Draw** menu, click **Pipe and Drape Wizard**.



2 Click the *AutoDrape* option button, and then click *Next*.

Result: The Define Drape Line Dimensions window appears.

3 To create a new section of drape, click *Create a New Drape Line*; to add a section to an existing drape panel, click *Add to an Existing Drape Line*.

4 In the *Length* box, type the total length of the drape line, or accept the default value of 40’0”.

5 In the Height box, type the height of the drape line, or accept the default value of 12’0”.

6 In the Drape Panels per Section box, type the number of drape panels that you want to see in each section of the drape line, or accept the default value of 3 panels per section. Each section is separated by a new, upright piece of pipe.

7 Click *Finish*.

Result: The wizard window closes and the drape line attaches to your cursor.

8 Click in the desired spot in your drawing to insert the drape line.

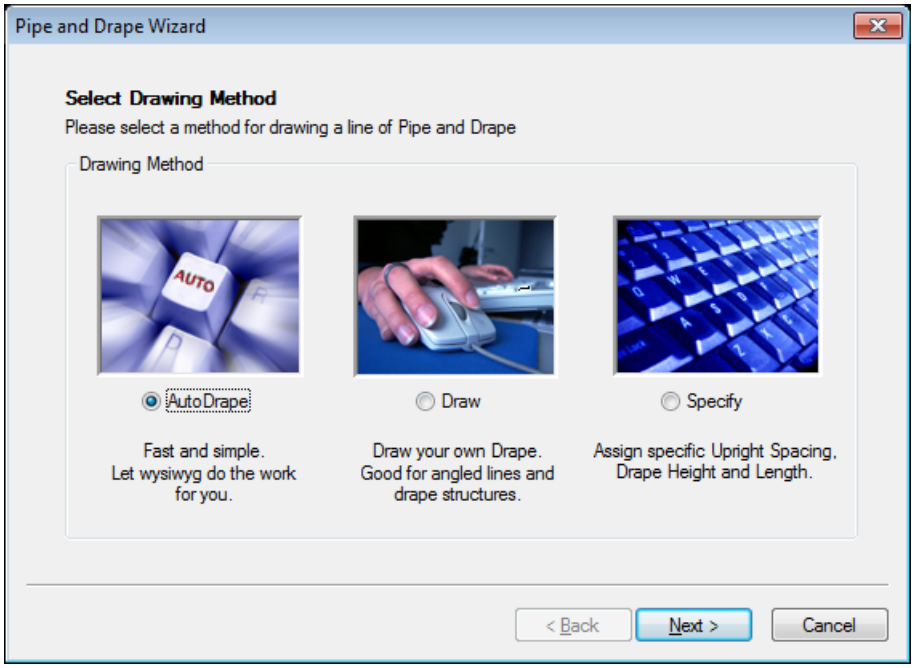
Note: For details on customizing the new drape line, see “To customize drapery elements (color, material, and texture)” on page 181.

To draw pipe and drape - Draw method

The Draw method is the most flexible way of adding pipe and drape because you can draw as many consecutive sections as you like, in any desired pattern. You can specify the height, maximum space between each upright pipe, and the number of panels per section, and then click and drag to draw as many sections as you like. After you draw the drape panel, you can further customize it by adding color, texture, and material to it.

Note: If you are adding a new drape section to an existing panel in your drawing, click the mid-point snap tool first. This ensures that the new drape panel "snaps" to the existing one in the correct position.

1 From the **Draw** menu, click **Pipe and Drape Wizard**.



2 Click the *Draw* option button, and then click *Next*.

Result: The Define Drape Line Dimensions window appears.

3 In the *Height* box, type the height of the drape line, or accept the default value of 12'0".

4 In the *Max. Upright Spacing* box, type the space between each upright piece of pipe, or accept the default value of 14'0". This is the *maximum* space between the upright pieces of pipe; however, if you click to insert a section at a shorter interval than this value, WYSIWYG automatically adjusts the width of each drapery panel accordingly and a shorter section is inserted in the drawing.

5 In the *Drape Panels per Section* box, type the number of drape panels that you want to see in each section of the drape line, or accept the default value of 3 panels per section. Each panel is separated by a new, upright piece of pipe.

6 Click the check box beside *Space Uprights Evenly* if you want WYSIWYG to automatically adjust the spacing of each drapery section so that the panels are of equal size. If you do not select this check box, then WYSIWYG creates drapery panels of equal size for all full-size sections, but resizes the panels in the last section accordingly if you choose to make this panel smaller.

7 Click *Finish*.

8 Click in the desired spot in your drawing where you want to start drawing the drape line.

9 Move the cursor to the spot where you want the first section to end, and click. Then move your cursor to the spot where you want the next section to end, and click. You can draw as many sections as you like. When you are finished drawing your drape line, right-click and select **Finish Pipe and Drape**.

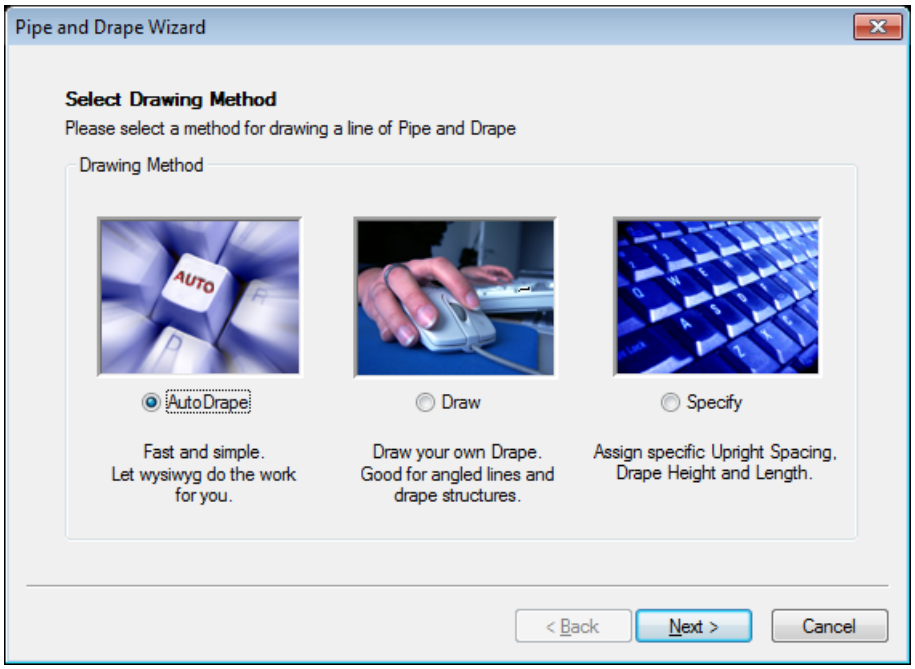
Note: For details on customizing the new drape line, see “To customize drapery elements (color, material, and texture)” on page 181.

To draw pipe and drape - Specify method

The Specify method lets you draw one section of drape line at a time of a size that you specify. You can choose the height, width per section, the number of sections, and the number of panels per section. After you draw the drape panel, you can further customize it by adding color, texture, and material to it.

Note: If you are adding a new drape section to an existing panel in your drawing, click the mid-point snap tool first. This ensures that the new drape panel “snaps” to the existing one in the correct position.

1 From the **Draw** menu, click **Pipe and Drape Wizard**.



2 Click the *Specify* option button, and then click *Next*.

Result: The Define Drape Line Dimensions window appears.

3 To create a *new* section of drape, click *Create a New Drape Line*; to add a section to an existing drape panel, click *Add to an Existing Drape Line*.

4 In the *Height* box, type the height of the drape line, or accept the default value of 12’0”.

5 In the *Width per Section* box, type the maximum space between each upright piece of pipe, or accept the default value of 14’0”.

6 In the *Number of Sections* box, type the number of sections in the drape line, or accept the default value of 1.

7 In the *Drape Panels per Section* box, type the number of drape panels that you want to see in each section of the drape line, or accept the default value of 3 panels per section. Each panel is separated by a new, upright piece of pipe.

8 Click *Finish*.

9 Click in the desired spot in your drawing where you want to start drawing the drape line.

10 Move the cursor to the spot where you want the first section to end, and click. Then move your cursor to the spot where you want the next section to end, and click. You can draw as many sections as you like. When you are finished drawing your drape line, right-click and select **Finish Pipe and Drape**.

Note: For details on customizing the new drape line, see “To customize drapery elements (color, material, and texture)” on page 181.

To quickly change Pipe and Drape Wizard settings

This procedure shows you how to quickly change some of the structural pipe and drape settings for a selected drape line, such as the height and diameter of the upright pipes and cross members. Use this procedure to customize the structural properties of a single drape line.

1 In your drawing, select the pipe and drape sections that you want to customize.To select multiple sections, press CTRL, and then click the appropriate sections.

Note: The sections that you select must have been drawn in the same way. For example, they must all be AutoDrape drape lines.

2 Right-click and select Change Pipe and Drape Settings.

Result: The appropriate Settings window appears. Based on the type of drape line that you selected, a different window appears.

3 Change any of the settings that you see in this window. For help on any of the boxes, click in the box, and then press F1.

4 Click *Finish* to save and apply your new settings.

To customize drapery elements (color, material, and texture)

After you add the required number of drape lines to your drawing, you can customize them by adding a custom color or texture to selected drapery elements (such as the front and back of the drape panels, the cross-members, upright pipes, the drape loops that attach to the cross-members, or the drape lines as a whole). You can also add a custom material to the drape line.

Notes:

- If you select multiple drape lines in your drawing, you can apply the same properties to all of them simultaneously through the following procedure. However, note that if you had already selected varying properties (for example, if you had chosen one color for the front panel of some drape lines, but another color for the front panel of others), or if some of the elements had no properties selected at all, then when you choose new properties in the following procedure, it applies to all drape lines selected, overwriting any existing settings that you may have saved.
- When you select multiple drape lines, the Appearance tab only lists the properties of selected elements if they are common across all drape lines. For example, if the front panel of all selected drape lines already has the same custom texture applied, and it is set to tile in one meter squared intervals, then these values are shown in the Appearance tab. However, if at least one of the front panels in the selected drape lines has a different custom texture value applied, the Appearance tab does not list the texture settings at all. If you choose new texture settings, your choice overwrites all existing settings for the selected element across all selected drape lines.
- You can apply the same properties to all drapery elements by highlighting “Group” in the Elements box.

1. In your drawing, select the pipe and drape sections that you want to customize. To select multiple sections, press CTRL, and then click the appropriate sections.
2. With the desired sections selected, right-click and select *Properties*.

   Result: The Properties dialog appears.
3. Click the Appearance tab.
4. In the left pane, click the drapery element that you want to customize. For example, to customize the front of the drape panels (each panel is double-sided), click Drape Front.

   Note: You can select multiple elements by pressing CTRL, and then clicking the elements; you can select *all* elements by clicking *Select All*.
5. You have a choice of applying *one* of the following settings to the selected elements:
   - *Use Layer Rendering Color*: Click this option to apply the same color as the current layer to the selected elements.
   - *Use Custom Color*: To apply a custom color, click this option button, and then click the color box. In the resulting color dialog, select the custom color that you want to use, or click *Define Custom Colors* to create your own color. Click *OK*, and then click *Apply*.
   - *Use Texture from Library*: Click this option to apply a custom texture file from the WYSIWYG library to the selected elements, and then click the ellipsis button (...). In the resulting dialog, navigate to and highlight the desired texture. Click *Select*.
   - *Use Texture from File:* To apply a custom texture file that you have created and saved in bitmap or jpeg format, click this option button, and then click the ellipsis button (...). Navigate to the file, and click *Open*.
6. If you have chosen to apply a texture to the elements, you can choose how the texture will appear on the selected drapery elements as follows (if you have applied a color instead, you can skip directly to step 7):
   - To view the texture file as a series of square or rectangular “tiles” that are repeated over the entire surface of the drapery element, click the *Tile* option button, and then type the size of the tiles in the boxes provided.
   - To view the texture file stretched out as a single surface over the entire drapery element, click the *Stretch* option button.
   - To preserve the aspect ratio of the texture so it is not distorted when stretched over the element, click *Preserve Aspect Ratio*.
   - To choose a custom border color for the texture when it is stretched over the surface of the element, click the *Border Color* box, and then choose the custom color.
   - To set the angle at which the texture appears on the drapery element, type the value in degrees in the *Rotation Angle* box.
7. To choose a custom material for the selected drapery elements (such as metal, paper, plastic, or fabric) click the ellipsis button (...) beside the *Material* box.
8. In the resulting dialog, navigate to and highlight the desired material, and then click *OK*.
9. Click *Apply*.
10. If desired, you can apply a transparency level to the selected element. Based on the settings that you have chosen for color, texture, and material, the transparency options vary as follows:
    - *Use Material Transparency:* All materials in WYSIWYG specify a level of transparency that varies from one material to the next. For example, glass is more transparent than brick. The transparency level of the selected material

appears beside this option button (with 0% being completely opaque). If you select Default, the transparency level is set to 0% (opaque). To customize the transparency level, click the Use Custom Transparency option button, and then type the desired transparency level in the box provided (or use the arrow to select a value).

- *Use Custom Transparency*: Click this option button if you want to set a transparency level for the color or texture that you have chosen, or if you want to override the default transparency level of the selected material. Then type the desired transparency level in the box provided (or use the arrow to select a value).

11 Click *Apply*.

12 Click *OK* to save your changes and close the Properties dialog.

To customize pipe and drape structure for individual drape lines

This procedure shows you how to customize other properties of the pipe and drape sections, such as the height and diameter of the upright pipes and cross members. Use this procedure to customize the structural properties of individual drape lines that you have selected.

Note: While the tabs in the Properties window are best used to customize a *single* drape line, if you have selected multiple drape lines, the values that you type in these tabs apply to *all* selected drape lines; they overwrite any custom settings that you may have previously saved for individual cross members, uprights, or drape panels.

1 In your drawing, select the pipe and drape section that you want to customize.

2 With the desired section selected, right-click and select **Properties**.

Result: The Properties window appears.

3 To change any properties of the drape line cross members, click the *Drape Cross Member* tab.

- *Length*: Type the length of all cross member pipes in the selected drape line.
- *Diameter*: Type the diameter of all cross member pipes in the selected drape line.
- *Use Document Defaults (Segments)*: Click to clear this check box if you want to specify the number of segments that comprise the circumference of the cross member pipes. You can choose between three and 40 segments. This value is largely for rendering purposes—the greater the number of segments, the greater the rendering time. Leave this check box checked to accept the default value of 15 segments.

4 Click *Apply*.

5 To change any properties of the drape line panels, click the *Drape Panel* tab.

- *Height*: Type the height of all panels in the selected drape line.
- *Width*: Type the width of all panels in the selected drape line.
- *Loop Diameter*: Type the diameter of the loops that wrap around the cross member pipes at the top of all panels in the selected drape line. This value should be greater than the diameter of the cross member pipes.
- *Use Document Defaults (Segments)*: Click to clear this check box if you want to specify the number of segments that comprise the drape panel loop that wraps around the cross member pipe. You can choose between three and 40 segments. This value is largely for rendering purposes—the greater the number of segments, the greater the rendering time. Leave this check box checked to accept the default value of 15 segments.

6 Click *Apply*.

7 To change any properties of the upright pipes, click the *Drape Upright* tab.

- *Height*: Type the height of all upright pipes in the selected drape line.
- *Diameter*: Type the diameter of all upright pipes in the selected drape line.
- *Use Document Defaults (Segments)*: Click to clear this check box if you want to specify the number of segments that comprise the circumference of the upright pipes. You can choose between three and 40 segments. This value is largely for rendering purposes—the greater the number of segments, the greater the rendering time. Leave this check box checked to accept the default value of 15 segments.

8 Click *Apply*.

9 Click *OK*.

Result: Your changes are applied to all selected drape lines.

BlackTrax Zone

There are three types of Zones can be defined: Rectangular Zones, Cylindrical Zones and Spherical Zones. These new CAD objects are used to define zones of interest for the BlackTrax system, which can be associated with a trigger relationship in the BlackTrax software.

The BlackTrax Zone objects must be given a unique name when created so they can be referenced in the BlackTrax System. By default, BlackTrax Zones will be appear as a red, semi-transparent volume in the Shaded Views.

Editing objects

Selecting

Before you can manipulate objects, you must select them. Selected objects are easily identifiable: a white box appears at the insertion point and the lines of the object become dotted. Fixtures that are selected are filled in green and their beams can be set to either on or off.

Use the following table to identify the selection method for objects.

Desired action	Selection method
Select a single object.	■ Click on the object.
Select multiple objects.	■ Press and hold CTRL while clicking on the desired objects. ■ Draw a box from left to right to select all objects fully within the box. ■ Draw a box from right to left to select all objects fully within the box, as well as the objects partially contained in the box.
Select all the objects on the current layer.	■ Press and hold CTRL+SHIFT and type *A* or use the *Select Current Layer* tool on the *Selection* toolbar. 
Select all the objects on the screen.	■ Press and hold CTRL and type *A* or use the *Select All* tool on the *Selection* toolbar. 
Select all the objects on one or more layers.	■ From the **Edit** menu, choose **Select Special**, and then choose **Specify layers to select**. You can also use the *Specify Layers* tool on the *Selection* toolbar.  ■ To select multiple layers, press and hold CTRL while selecting the layers.
Set beam display for selected fixtures.	■ Set the *Show beams for selected fixtures* option. For more information on this option, refer to the “Show Details tab” on page 76.
Select all instances of a particular library item.	■ In Wireframe view, right-click and choose **Select Library Items**, and then either select the library item from the menu that appears, or click **Specify** to select the item from a selection box.
Select all library items.	■ In Wireframe view, right-click and choose **Select Library Items > Select All Library Items.**

Desired action	Selection method
Select all instances of a particular fixture.	■ In Wireframe view, right-click and choose **Select Fixtures**, and then either select the fixture from the menu that appears, or click **Specify** to select the fixture from a selection box. To save time, you can also press SHIFT+S.
Select all fixtures.	■ In Wireframe view, right-click and choose **Select Fixtures > Select All Fixtures**.
Select all fixtures on a particular type of pipe or truss.	■ In Wireframe view, select the type of pipe/truss containing the fixture that you want to select. Right-click and choose **Select Fixtures > Select on Selected Pipe/ Truss** and then choose the type of fixture from the list, or click **Select All** to select all fixtures on this type of pipe or truss.
Select fixtures by their fixture group.	■ In CAD Mode (Wireframe, Quad, Shaded View), DATA Mode (H Select and V Select), DESIGN Mode (Wireframe, Quad, and Shaded View), and LIVE Mode (Wireframe, Quad, and Shaded View), right-click to select fixtures by their fixture group.
Select fixtures by their lighting position.	■ In CAD Mode (Wireframe, Quad, Shaded View), DATA Mode (H Select and V Select), DESIGN Mode (Wireframe, Quad, and Shaded View), and LIVE Mode (Wireframe, Quad, and Shaded View), you can right-click to select fixtures by their lighting position. Note, however, that you have to name all of your lighting positions before you can use this feature. If you do not have any lighting positions named, then you cannot use this tool because the position names will not populate in the list.

To select various fixtures in your file, right-click in a wireframe, click *Select All*, and then choose which fixture type to select. This functionality is available in CAD, Design, and Live modes.

Cycle Selection

This feature is helpful when you need to select an object that shares an edge with another object. For example, if you need to select the roof of a building that shares its edges with the building's walls. Left-to-right drag-selection in a side or elevation view works, of course, but there are many cases when this method cannot be used.

To use cycle selection

To activate Cycle Selection, simply press SHIFT before clicking to select. When Cycle Selection is active, *Cycle On* appears in the WYSIWYG Status Bar.

To select the object, click in a spot where two or more lines overlap or intersect (“lines” do not necessarily mean Line objects, but rather lines that are part of anything drawn or inserted from the library).

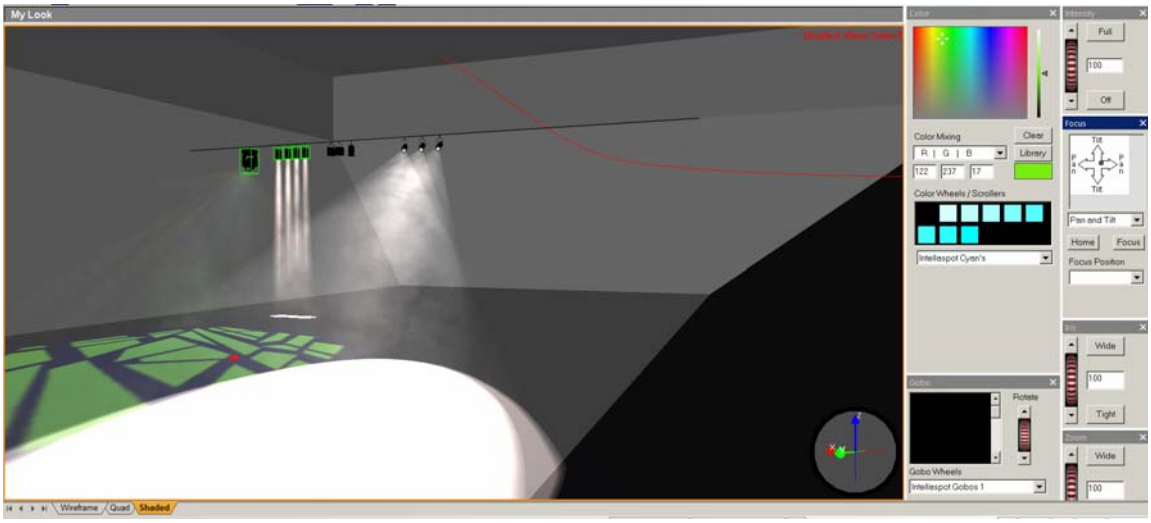
To continue selecting, keep pressing SHIFT to continue cycling through all the items that share the coordinate where you have clicked (in the order in which they were added to the drawing).

Tips:

- If you also press CTRL at the same time as SHIFT, you can add items to an existing selection set when those items can only be selected by Cycle Selection.
- Hold down CTRL+SHIFT and click at a coordinate where something in the same “cycle set” was already selected to cycle through all items again, including the one already selected.
- Ensure that when cycling through the cycle set, you do not by mistake deselect an item that you need to have selected (although in order to do so, you would have to release CTRL+SHIFT when you cycle to this item instead of holding CTRL+SHIFT down as you “pass over it”).

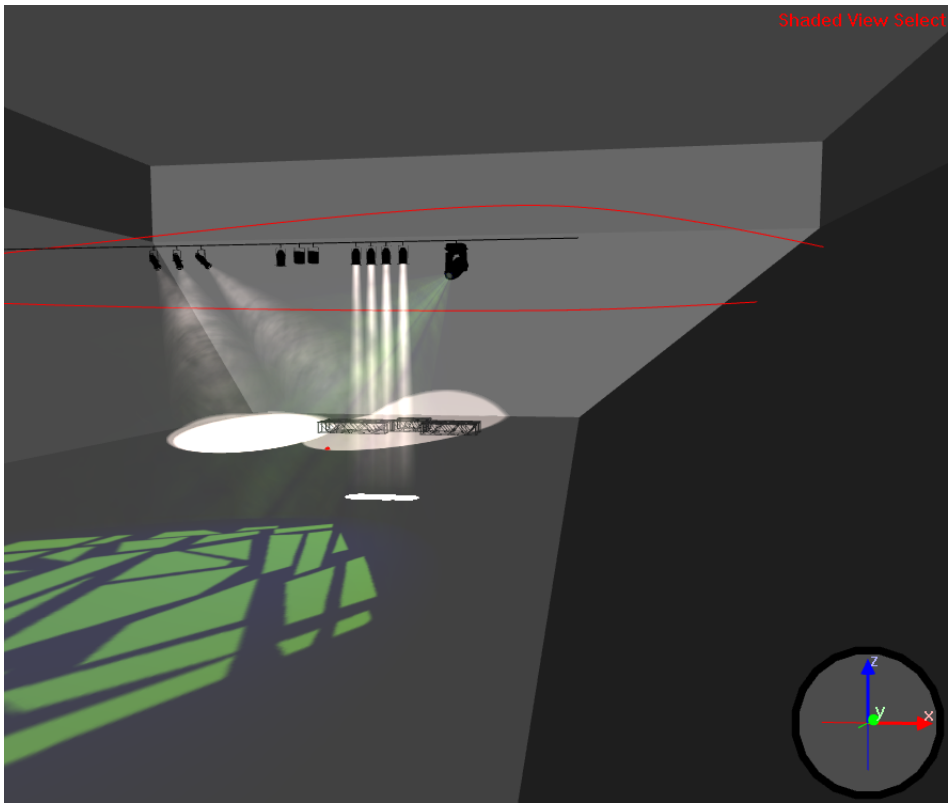
Shaded View Selection

You can select fixtures in Shaded view in CAD, Design and Live modes. This allows you to click on a fixture in the Shaded view, which will then activate the fixture for editing with the Designer tools. You can select multiple fixtures at one time by either drag-selecting or holding down CTRL while selecting fixtures.



Using Shaded View Selection

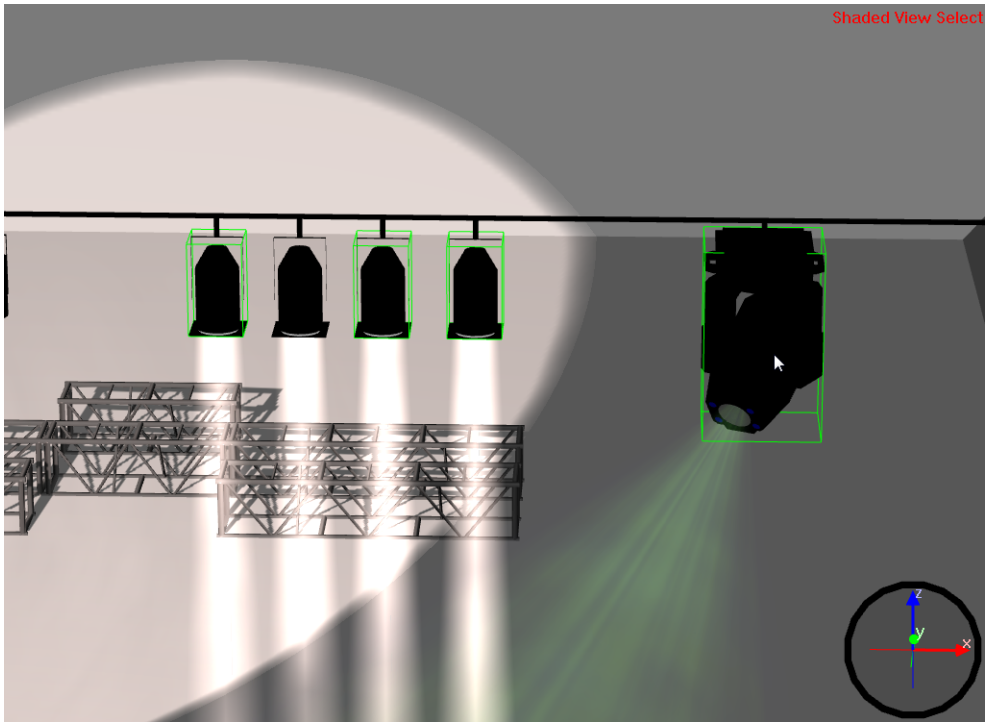
You can activate Shaded View Selection by pressing TAB while in CAD, Design or Live mode > Shaded View. Once you press TAB, you will notice a red message in the top right-hand corner of the Shaded view window: “Shaded View Select ON”.



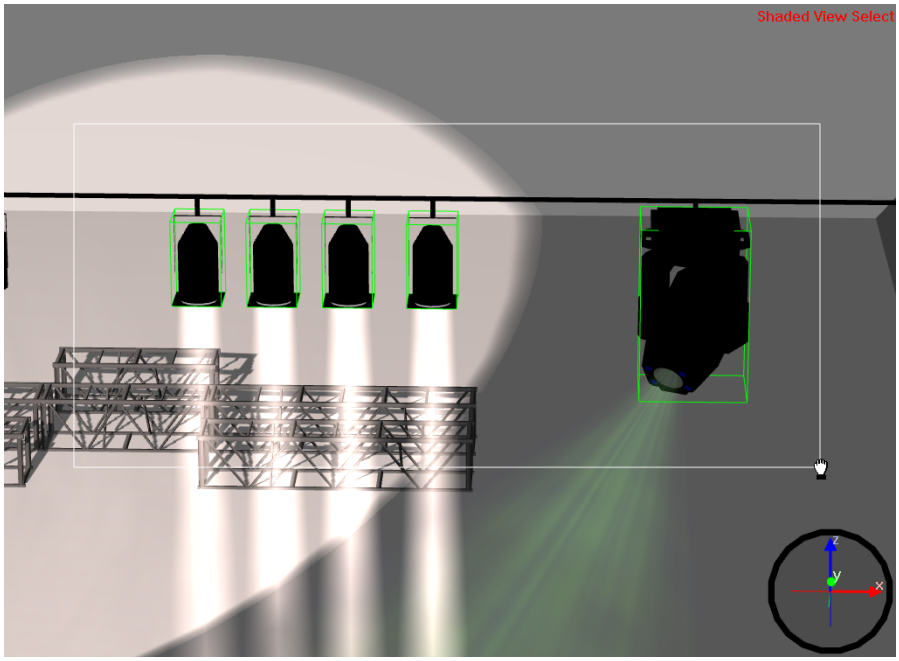
To Activate Shaded View Selection

1 In a Shaded view in CAD, Live or Design mode (meaning the amber outline appears around the Shaded view), press TAB once. You are now in Shaded View Selection mode.

2 To select a fixture in Shaded View Selection mode, click directly on a fixture's image in the Shaded view:



3 To select multiple fixtures in Shaded View Selection, press CTRL and click each fixture that you want to select. You can also select multiple fixtures at once by drag-selecting with the mouse as you would in wireframe modes; Shaded View Selection disables the camera movement controls so you can drag a marquee box around fixtures to select them.



4 Once you are finished selecting fixtures, press TAB again to exit from Shaded View Selection mode. You will then regain control of the Shaded View camera.

Tips:

- It is helpful to keep your hand near the TAB key while selecting fixtures and maneuvering around with the camera; once you need to move around your view, press TAB to exit Shaded View Selection. Your fixture selection is maintained as you move in and out of the Shaded View Selection mode.
- Make sure while you navigate around in the Shaded View that you give yourself a good angle on the view in which you want to select fixtures. It may be difficult to select a fixture in the middle of a lighting position or truss while you're looking at a side view of it.

To focus a fixture using Shaded View Selection

Shaded View Selection enables you to focus your fixture by clicking on the target in Shaded views.

1 Activate Shaded View Selection by pressing TAB while in Design or Live mode > Shaded View.

Result: A red message appears in the top right-hand corner of the Shaded view window: "Shaded View Select ON".

2 Click to select the fixture that you want to focus. To select multiple fixtures, press CTRL and click to select them. Alternately, fixtures can be selected using the Fixture Selection tool bar.

3 Activate the focus feature by pressing F on your keyboard (or right-clicking and selecting **Focus** from the menu). Alternately, you can press L to activate the "Full and Focus" feature (or right-click and select **Full and Focus** from the menu). The status bar will indicate when a focus operation is active.

4 Click in the Shaded view to select the focus target.

5 Press TAB to exit Shaded View Selection.

Using the Fixture Selection toolbar

The Fixture Selection toolbar offers two convenient methods to select fixtures in your file and a method to traverse your selection.



The toolbar has a combo box in which you can type in Fixture spot numbers that you would like to select (as you would on most lighting consoles). For a consecutive range of fixtures, use a dash (-), or to enter a list use commas (,)

Note: This method will only work if fixtures have already been assigned Spot numbers.)

Example: "1-10, 15, 20" will select 12 fixtures, with Spot numbers: 1,2,3,4,5,6,7,8,9,10,15,20.

The combo box will remember previous selection strings and display them when you click on the down arrow.

Next, the Fixture Selection toolbar offers a drop-down menu of all the "Select Fixture By" methods listed in the menus.

Use the Next button to traverse the fixtures selected one by one from first to last, or use the Previous button to traverse the fixtures selected one by one in the reverse order. Select the All button to select all fixtures in your current selection.

Undo/Redo

If you discover that you have made an error while editing an object, you can easily reverse the error by using the Undo tool available in WYSIWYG. The name of the Undo tool changes based on what the last action was (for example, if the last action was a Paste operation, the Undo tool will be Undo Paste). You can use Undo to reverse the last series of actions until the tool is greyed out.

If you decide that you did not want to undo an action, use the Redo tool. Similar to the Undo tool, the name of the Redo tool changes based on what the last undo action was.

To use the Undo tool

From the **Edit** menu, choose **Undo**.

Tip: You may also use the *Undo* tool on the *Edit* toolbar. 

Result: The last action you performed is reversed.

To use the Redo tool

From the **Edit** menu, choose **Redo**.

Tip: You may also use the *Redo* tool  on the *Edit* toolbar.

Result: The last undo action you performed is reversed.

Grouping / ungrouping

Grouping objects together ensures that they will always be selected as one. For example, you may want to group an object and its corresponding focus position. If you relocate the object in the drawing, the focus position will automatically follow, as will all the light beams pointing at it, saving you time and work.

To group objects

1 Select the objects to be grouped.

2 From the **Edit** menu, choose **Group**.

Tip: You may also use the *Group* tool on the *Tools* toolbar. 

To ungroup objects

1 Select the grouped objects.

2 From the **Edit** menu, choose **Ungroup**.

Tip: You may also use the *Ungroup* tool on the *Tools* toolbar. 

Note:
You can group together objects located on different layers, but the resulting group can only exist on one layer.

To change or view the layer of a group

1 Select the object(s) for which you want to view the grouping.

2 Right-click and select **Properties**.

Tip: At any time, to access an object's properties, you can click on the *Properties* tool on the *Edit* toolbar. 

3 Click the *Group* tab.

Result: The layers for the objects in the group are listed with the layer for the group highlighted.

Note: By default, the group assumes the current layer when initially created.

4 To group the objects on a different layer, select the layer from the list.

5 Click *OK*.

Moving

WYSIWYG supports two types of move commands: relative and absolute.

- Absolute coordinates are relative to the origin of the drawing. An absolute move is defined as moving an object from one specific point to another.
- A relative move is defined as moving an object to a destination relative to its original position.

To move an object using the mouse

1 Select the objects to be moved.

2 Hover the cursor over the object until a grey box appears, and then click and hold.

3 Drag the object to the new position, and then release the mouse button to finish the move.

To move an object using absolute values

1 Select the objects to be moved.

2 From the **Edit** menu, choose **Move**.

Tip: You may also use the *Move* tool on the *Tools* toolbar.

3 Press ENTER on your keyboard to move the object from its insertion point, OR click to set the base point for the move, OR type the coordinates of the base point, and then press ENTER.

Note: When you start typing, a box appears at the bottom left-hand corner of the window, showing the value that you enter.

4 Type the new coordinate location for the base point, and then press ENTER.

To move an object using relative values

1 Select the objects to be moved.

2 From the **Edit** menu, choose **Move**.

Tip: You may also use the *Move* tool on the *Tools* toolbar.

3 Type **@**, followed by the distances to move the object, and then press ENTER. For example, to move an object 2’-0” stage right (X axis) and 3’-0” off the floor (Z axis), type the following, and then press ENTER:

@2’,0,3’

To move an object using distance and direction

1 Select the objects to be moved.

2 From the **Edit** menu, choose **Move**.

Tip: You may also use the *Move* tool on the *Tools* toolbar.

3 Type **@**, the distance that you want to move the object, the < sign, and the direction (as an angle, positive or negative) to move the object, and then press ENTER. For example, to move an object 3’-6” to the left, type the following, and then press ENTER:

@3’6”<-180

Nudging

To save time and for added precision when working in your model, you can use the Nudge feature to nudge any object except fixtures. The amount of the nudge is equal to the size of the Grid. For information on changing the grid size, refer to “Draw Defaults tab” on page 404.

To nudge an object

1 In your model, click to select the object that you want to move.

2 Press and hold the SHIFT and ALT keys on your keyboard simultaneously, and then press the ARROW keys on your keyboard to nudge the object in the desired direction.

Tip: To nudge objects in increments ten times larger than the size of the grid, do not hold down SHIFT.

Sending to Back / Bringing to Front

The feature is very useful for correcting draw order issues (among other things). WYSIWYG displays objects “on top of each other” in the order in which they were added to the drawing. For example, if you draw/insert a table after you draw/insert a piece of truss above it (i.e., at a different missing coordinate), the table will appear “above” the truss (in Plan View), even though the table is on the floor and the truss is up in the air.

You can use the *Send to Back* or *Bring to Front* commands to arrange the way the objects appear in your drawing. Note, however, that when you do so the physical location of the objects is unchanged; you are only changing the way in which they are displayed.

To send an object to the back / bring an object to the front

Notes:

- The *Send to Back* option will “place” the selected object “below” everything else in the drawing.
- The *Bring to Front* option will “place” the selected object “above” everything else in the drawing.
- You may note that sometimes the object will not move from its current position. This may be due to the *Draw Order* settings on the *Draw Options tab* of User Options and View options (for details, see *Draw Order* on page 405). These settings supersede any Send/Bring commands by essentially telling WYSIWYG to *always* draw fixtures, objects, and hang structures in the order shown.

1 In CAD mode, select the object that you want to send to back or bring to front.

2 Right-click and select **Bring to Front** or **Send to Back**.

Tip:

Alternately, you can use hotkeys to send items to the back/bring them to the front:

- open curly bracket for Bring to Front (i.e., **{** or **SHIFT+[**)
- closed curly bracket for Send to Back (i.e., **}** or **SHIFT+]**)

Rotating

You can rotate objects around a base point to place them on angles in your drawing.

To rotate an object

1 Select the object(s) to be rotated.

2 From the **Edit** menu, choose **Rotate**.

Note: You may also use the Rotate tool on the Tools toolbar.

- Press *ENTER* on your keyboard to rotate the object from its insertion point.

OR

- Click to set the center point for the rotation.

OR

- Type in the coordinates of the center point for the rotation, and then press *ENTER*.

3 Type the rotation angle, and then press *ENTER*, or drag the mouse and click to type the rotation angle. For example, to rotate an object 45 degrees, type 45, and then press *ENTER*.

Rotating in Place

You can rotate multiple objects around their respective insertion points to place them all at the same angle in your drawing.

To rotate multiple objects in place

1 Select the objects to be rotated.

2 From the **Edit** menu, choose **Rotate in Place**.

- Click to set the base point used for the rotation angle.

OR

- Type in the coordinates of the base point, and then press *ENTER*.

Note: The base point is only used to help determine the angle, as each object is rotated around its own insertion point.

3 Type the rotation angle, and then press *ENTER*, or move the mouse and click to set the rotation angle. For example, to rotate objects 45 degrees, type 45, and then press *ENTER*.

Mirroring

The Mirror command duplicates and reverses an object, inserting it the same distance from an axis line as the original object.

Note: After mirroring an object that was originally created through a merge operation, you cannot reliably perform another merge procedure on this object; therefore, it is recommended that you perform all merge operations on these objects *before* you mirror them.

To insert a mirrored object

1 Select the object(s) to be mirrored.

2 From the **Edit** menu, choose **Mirror**.

Tip: You may also use the *Mirror* tool on the *Tools* toolbar. 

3 Click on the drawing to enter the first axis point.

4 Click on the drawing to set the second axis point.

Result: The mirrored object is copied, reversed from the origin, based on the axis drawn.

Modifying object shapes

You can change the shape of certain objects after they are drawn. The objects that you can change include lines, risers, cameras, surfaces, spheres, and arcs. Modifying the shape changes one or more of the parameters of the object, such as its width, height, or radius.

Note: If you are changing the shape of a sphere, when you click and drag one of its vertices, it will be resized proportionately if the *Lock ratio* checkbox is checked for the sphere's properties. (Click *Properties* > *Sphere* tab, and then ensure the *Lock ratio* check box is checked.) To resize the sphere in a disproportionate manner, deselect this checkbox, and then click and drag one of its vertices.

To modify the shape of an object

1 Select the object whose shape you want to modify.

Result: You will notice that several white boxes will appear at the vertices of the object. These boxes are referred to as “markers.”

2 Move your cursor to a vertex.

Result: The cursor changes to an arrow.

Note: If you see a grey dotted box beside your mouse cursor, the object will be moved instead of reshaped. Move your cursor so that only the arrow appears.

3 Left-click to “grip” one of the markers.

4 While holding down the left mouse button, click and drag the object until it reaches the desired shape.

5 Release the mouse button to set the shape.

Tip: As you move the mouse, the current coordinates appear at the bottom of the screen.

Object properties

All objects in WYSIWYG have properties. Properties are the characteristics or attributes that shape and define an object. Modifying object properties is a form of editing and is the only way to make changes to objects once they have been drawn (some exceptions exist).

To edit object properties

1 Select the objects that you want to edit.

2 Right-click on the object, and then select **Properties**.

Tip: At any time, to access an object’s properties, you can click on the *Properties* tool on the *Edit* toolbar. 

Result: The *Properties* dialog is displayed.

Note: All objects have general properties that affect the color and layer on which they are drawn. Objects also have properties that are particular to that type of object. For example, fixtures have unit numbers but do not have a radius. When an object is selected and its properties are displayed, a tab appears in the properties dialog box for that type of object. When you select multiple objects of different types, tabs appear for each type of object selected.

General object properties

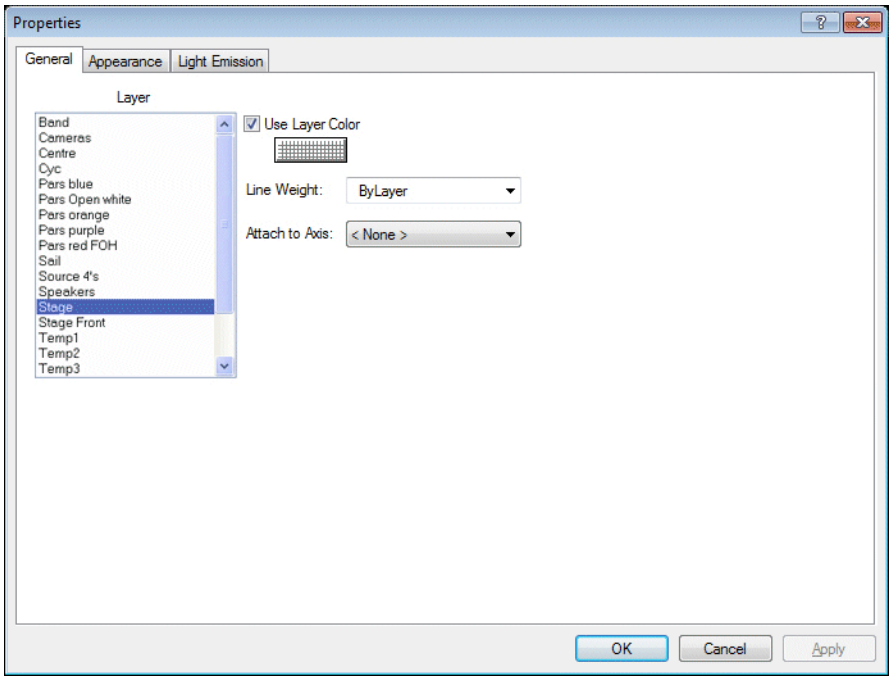
In WYSIWYG, there are four tabs that appear in the properties dialog box for every object. These four tabs are the *General*, *Appearance*, *Light Emission*, and *Sidedness* tabs.

For 3D solids and surfaces, the *Set Piece* tab also appears in the properties dialog box.

All tabs are explained here.

General tab

Options on the *General* tab affect the layer’s color and line weight.



- *Layer* list: A list of the layers in your document is displayed. Click on a list item to change the layer on which the objects will be drawn.
- *Use Layer Color*: Select this check box to set the object's color for wireframe views to be the same as the properties of the layer on which the object resides. For more information, refer to "Layer properties" on page 220.

  Click to clear the check box, and then use the color button to set a specific color for the object.
- *Line Weight*: From the drop-down list, choose the specific line weight for the selected object. The applied line weight is visible in all Wireframe views in all modes. For objects only, you can also choose the default setting, *ByLayer*, which sets the object's line weight to be the same as the properties of the layer on which the object resides. The value *Default* means that the document's line weight setting from the Show Options > Object Settings window is applied to the object or layer. For more information, refer to "Layer properties" on page 220.
- *Attach to Axis*: From the drop-down list, select the motion axis to which you want to attach the selected object or objects. For details on attaching objects to motion axes, see "To attach an object to a motion axis" on page 172.

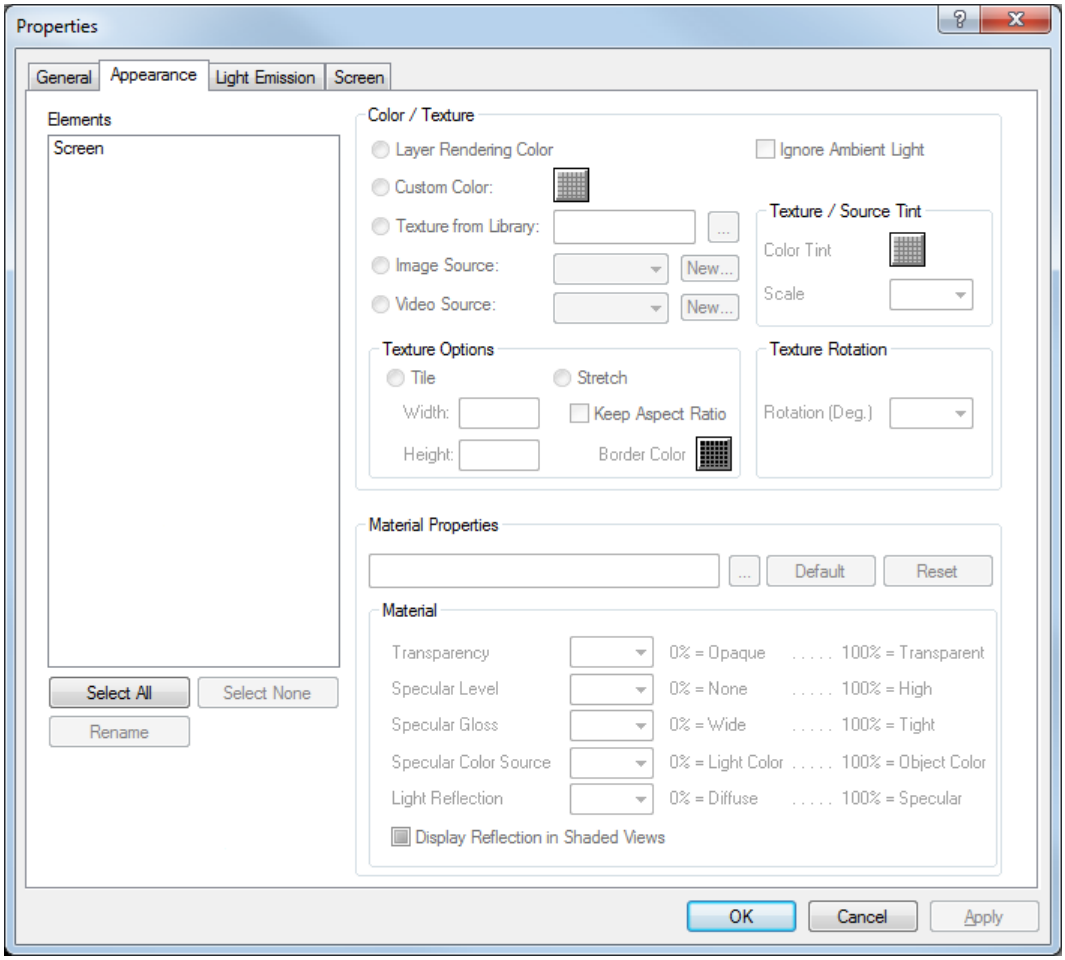
Note: You cannot attach Camera paths nor Cameras to axes. To create a moving Camera, you can either draw a Camera Path or use the DMX Camera.

Appearance tab

Use this tab to specify the appearance of the different components that make up the currently selected items. Based on the items that you have selected, the options in this tab differ.

You can use this tab to customize objects by adding different materials, colors, or textures to each element of the item. For example, if you have selected a library object, such as a podium, you can apply different materials, colors, and textures to each part of it (the body and the top portion).

You can also use this tab to rename the individual elements of the currently selected object. This is useful for objects that contain many elements with similar names, like risers.



Note: You can apply textures to venues, surfaces, risers, walls, library items, spheres, cylinders, and cones. You can apply transparency to everything. When customizing the selected element, you can choose between applying a custom color or a custom texture—you cannot choose both. In addition to either color or texture, you can also apply material.

Highlight the element that you want to customize, and then choose one of the following options:

- *Ignore Ambient Light:* If checked, the selected element(s) will always appear in Shaded Views using the color values derived from the Appearance options specified (e.g., texture). While light emission and light from fixtures will further saturate the colors displayed for the selected object, ambient light will never affect this object if this option is enabled. This option is useful when objects (except Screens) are textured with images or video, and these need to appear in their original colors at all times. In renderings, this setting is ignored if light emission is enabled.
- *Layer Rendering Color:* Check this option to set the object(s) color for shaded views and renderings to the same as the properties of the layer on which the object resides. For more information on layer properties, refer to “Layer properties” on page 220.
- *Use Custom Color*: Click this option and then use the color button to set a specific color for the object(s).
- *Use Texture from Library*: Click this option to apply a texture from the library to the selected object(s). In the resulting window, navigate to and select the desired texture. You can apply textures to venues, surfaces, risers, walls, library items, spheres, cylinders, and cones. Click the ellipsis button (...) to change the selected texture file.

- *Use Image Source*: Click this option to apply to the selected object(s) a texture that you have created and saved in either .bmp, .jpg, .png, or .gif format. To apply the image, you must create a new Image Source. To do so, click *New*, and then use the Image Manager to create a new Image Source with the image you created. For more information about creating Image Sources, see “To create a new image source in the image manager” on page 344.
- *Use Video Source*: You can apply a video source or a subsource as a "dynamic texture” to 3D primitives--surfaces, risers, spheres, cylinders, or cones--or to individual elements of 3D primitives. Select this option to apply a previously created video source or subsource, or click *New* to create a new video source or subsource directly from this window. For details on this feature, see “Video Manager” on page 335.

If you have chosen a texture option, then you can also set the properties of the texture as follows:

- *Tile*: Click this option button if you want to have the texture repeated over the selected element in a continuous series of squares or rectangles, and then type the size of the frame in which you want the texture to appear in the *Width* and *Height* boxes. Based on the size that you enter, WYSIWYG calculates how many times the texture is repeated (or tiled) to completely cover the selected element.
  Note: When tiling a texture on a sphere, cylinder, or cone, by default it will completely wrap around the object. If you change the tile size, you will scale the texture up or down accordingly. If you revert back to the original tile values, you return to the default view.
- *Stretch:* To have the texture stretch over the entire element surface, click this option button. Based on the aspect ratio and the rotation angle, WYSIWYG evaluates the surface with all of its edges and stretches the texture so the best fit is used. Note that this option is not applicable when you apply textures to spheres, cylinders, or cones. For these objects, *Tile* is the only option.
- *Keep Aspect Ratio*: Click this option to keep the aspect ratio of the original image when it is stretched over the surface of the object you have selected. This option helps avoid distortion of complex textures. If the texture image cannot wrap completely over the entire surface of the object while maintaining its original aspect ratio, then the color that you choose in the *Border Color* box will be applied evenly around any excess surface area not covered by the texture (much like a picture frame around a picture).
- *Border Color*: If you have chosen to preserve the texture’s aspect ratio, click this box to choose the color that will be applied evenly around any excess surface area not covered by the texture (much like a picture frame around a picture).
- *Texture / Source Tint*: This option will allow the color of an object’s texture to be altered from the source color.
- *Texture Rotation:* If the texture has text or another recognizable image in it, you might need to rotate the image to get it right-side up. Select the rotation angle from this box.

Proceed with the following properties settings:

- *Material*: Click the ellipsis button (...) to choose a material for the selected elements. Click *Default* to remove the selected material from the element and return to the default material. For more information on materials, refer to “Materials” on page 213.
- *Material Properties:* When a material is selected, the properties of the material will vary from one material to the next. The Material Properties section enables these default material settings to be changed. Each property value can be changed on a scale from 0%-100%. An explanation of the value is found on the right side of the property.

- *Transparency*: This setting controls the proportion of light that passes though the material. 0% being completely opaque, 100% being completely clear.
- *Specular Level*: This setting controls how prominent other specular effects appear on the material. 0% being no visibility of specular effects, 100% being a maximum visibility of specular effects.
- *Specular Gloss*: This settings controls the level of gloss a material will project. 0% will project the gloss over a wide area, 100% will concentrate the gloss in a small area.
- *Specular Color Source*: This setting controls the color of light projected on a material. 0% will show only the color of the light, 100% will show only the color of the material.
- *Light Reflection*: This setting controls the amount of light that is reflected off the material. 0% having no reflection, 100% being specular.
- *Display Reflection in Shaded Views*: Will display reflections in Shaded Views when enabled. For more information see “Material Reflections” on page 216.

To rename object elements

You can use this procedure to rename object elements that have similar names, giving them a more descriptive name. For example, if you have customized a riser, you could give each side of the riser a distinct name according to its custom properties. This is especially useful for custom objects that you have drawn and are going to save as custom library items. Since you cannot rename individual elements of custom library items, it is best to do so before you save the item.

1. Under Elements, highlight the element that you want to rename. For example, highlight *Riser - Back*.
2. Click *Rename*.
3. Type the new, descriptive name for the element.
4. Click *OK*.
5. Click *OK* in the Appearance tab to save your changes.

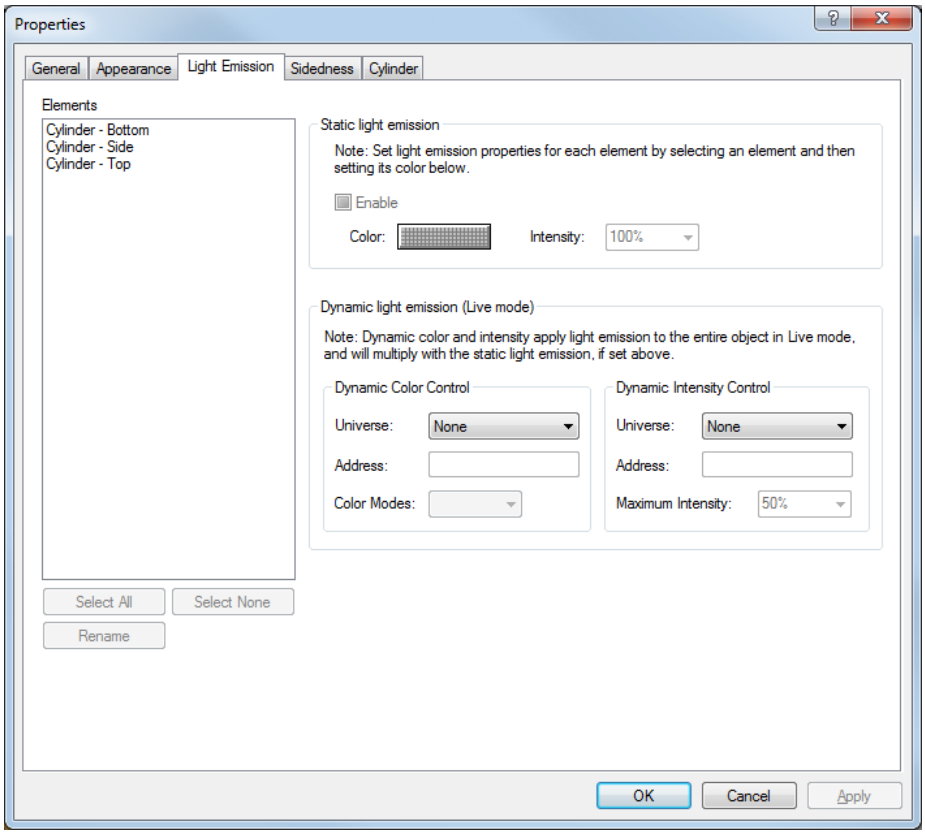
Light Emission tab

Options on the *Light Emission* tab affect the light emission properties for the selected object(s) or the elements that make up an object. Light emission is defined as the ability for objects to glow or emit light and can be used to simulate LEDs, lasers, neon, light boxes, projection screens, and so on. Light emissions can be either Static or Dynamic.

Static light emissions can be seen in **CAD**, **DESIGN** and unpatched **LIVE** mode. With Static light emissions you can choose different light emissions for each element in an object.

Dynamic light emission can be seen in patched **LIVE** mode. With Dynamic light emissions each element of the object will inherit the same light emission properties.

Note: Instead of using the *Properties* > *Light Emission* tab to change the light emission for the selected object, you can use the Quick Light Emission Tool. For details, see “Quick Light Emission Tool” on page 201.



Highlight the element that you want to customize, and then choose one of the following options:

- *Static light emission:*
  - *Enable Light Emission*: Click this box to enable Static light emission for the object, and then choose the light emitting properties.
  - *Color*: Click the color box to choose the static light-emitting color for the object. If you set only this value, and not the Dynamic Color Control, then this is the light-emitting color that the object will have in all shaded views and renderings in all modes (CAD, Design, Live). However, if you set the dynamic color value as well, then you can control the light-emitting color of the object in Live mode when connected to a console. (The static value still applies in Design mode.)
  - *Intensity*: Choose the static intensity of the light emission for the selected object. If you set only this value, and not the Dynamic Intensity Control, then this is the maximum light-emitting intensity that the object will have in all shaded views and renderings in all modes (CAD, Design, Live). However, if you set the dynamic intensity value as well, then you can control the light-emitting intensity of the object in Live mode when connected to a console. (The static value still applies in Design mode.)
- *Dynamic light emission (Live mode)*:

  Note: Dynamic light emission will apply to all elements of an object.
  - *Dynamic Color Control - Universe*: When you select a value from this drop-down box, the selected object will have light-emitting properties controllable by three DMX channels (red, blue, green). The object must be patched according to the WYSIWYG patch notation: *universe.DMX address*. From the drop-down box, select the patch universe of the selected object, and then type the starting DMX address in the *Address* box below it. If you set this value, it overrides the static color control that you chose above (if any) in Live mode when you are connected to a console.
  - *Dynamic Color Control - Address*: After selecting the patch universe of the selected object, type the starting DMX address for the patch in this box.

- *Color Mode*: Component color settings are designed to be used in groups of three, meaning that you choose the color mode for one of the surfaces and then continue choosing color modes for the other two surfaces in the triplet, assigning one color to each surface (either red, green or blue). If you are not selecting colors in a group of three, choose *Full Color Mode* in this drop-down box.

  This feature is particularly useful to emulate a color-mixing source with discrete red/green/blue elements, such as fluorescent tubes or LEDs. The color mode affects how the surface responds to the colors that you assign to it, both static and dynamic. In Full Color mode, the surface will show the resultant color mix of the RGB channels controlling it. For example, if you choose white in the color picker (RGB 255, 255, 255), the surface will be white. However, when you create a triplet surface using each component color, one for each surface, and then you choose one of the component colors, such as *Component Color - Red*, then this surface will only show the red component of the color.
- *Dynamic Intensity Control - Maximum Intensity*: Enables you to set the maximum intensity of patched objects. Select the intensity from the drop-down box.

Note: The DMX intensity value is treated as a percentage of the dynamic intensity value. For example, a DMX value of 127 results in 50% of the maximum intensity value that you set.

- *Dynamic Intensity Control - Universe*: When you select a value from this drop-down box, the selected object will have light emitting properties to a *maximum* of the intensity that you specify. The object must be patched according to the WYSIWYG patch notation: *universe*.*DMX address*. From the drop-down box, select the patch universe of the selected object, and then type the starting DMX address in the *Address* box below it.
- *Dynamic Intensity Control - Address*: After selecting the patch universe of the selected object, type the starting DMX address in this box.
- *Dynamic Intensity Control - Maximum Intensity*: Enables you to set the maximum intensity of patched objects. Select the intensity from the drop-down box.

Quick Light Emission Tool

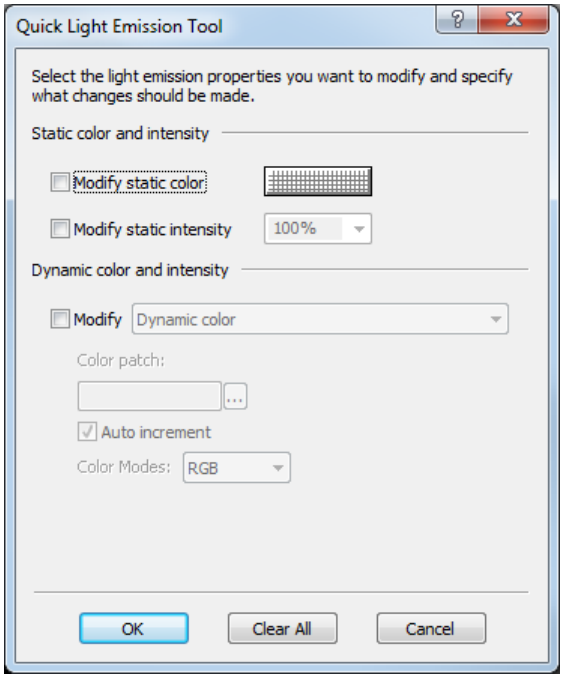
Instead of using the *Properties* > *Light Emission* tab to change the selected object's light emission, you can use the Quick Light Emission Tool. These tools allow you to quickly assign or change light emission properties for the selected object without opening the Properties window. You can enter repetitive information, as well as incremental information for each object as you click on it.

Note: The Quick Light Emission tool applies light emission properties to the entire object. All elements of the object will be assigned the same color when using the Quick Light Emission tool.

To use the Quick Light Emission Tool

1 From the **Tools** menu, choose **Quick Light Emission Tool**.

Tip: You can also click the *Quick Light Emission Tool* from the *Tools* toolbar. 



Select the light emission properties for the selected object. For details on each of the options, see “Light Emission tab” on page 199. If you want the value to auto increment for color, make sure you select the *Auto Increment* check box. WYSIWYG will assign the next sequential number based on the properties and requirements of the previous patch.

2 When you have chosen all desired values, click *OK*.

Sidedness tab

Options on the *Sidedness* tab affect how the selected object appears in Shaded views and Renderings. You cannot change the sidedness of any of the objects that come with WYSIWYG, either library objects or default venues. By default, all objects in the WYSIWYG library are double-sided, which means that when you rotate them in Shaded views, all sides display equally. Conversely, the default venues that come with WYSIWYG are single sided, which enables you to see “into” the venue when you rotate it in Shaded views.

You can, however, change the sidedness of objects that you have drawn in another program and imported into WYSIWYG (i.e., .dwg, .dxf files or SketchUp files) or objects that you have drawn in WYSIWYG, such as custom venues, surfaces or custom objects that you have added to the library.

For these objects, you can use this tab to change the sidedness from single to double sided, or vice versa. For example, if you have drawn a venue as double sided, you can select it and change it to single sided so that it behaves in the same manner as the default WYSIWYG venues (i.e., you can see into the venue as you rotate it in Shaded views). If you prefer to have an outside view of the custom venue, then the double-sided option is best as it prevents you from seeing “through” the walls. You can also use this feature to flip the faces of a custom surface from one direction to the other if you do not like the way the object appears in the Shaded view.

To change an object's sidedness

You can use this procedure to change objects from single to double sided and vice versa. You can also flip the faces of a single-sided object so that they are oriented in the opposite direction, either inward or outward.

Note: You cannot change the sidedness of any of the objects that come with WYSIWYG, either library objects or default venues. By default, all objects in the WYSIWYG library are double-sided, which means that when you rotate them in Shaded views, all sides display equally. Conversely, the default venues that come with WYSIWYG are single sided, which allows you to see "into" the venue when you rotate them in Shaded views.

1 Select the object that you want to edit.

2 Right-click on the object, and then select **Properties**.

Tip: At any time, to access an object's properties, you can click on the *Properties* tool on the *Edit* toolbar. 

Result: The *Properties* dialog is displayed.

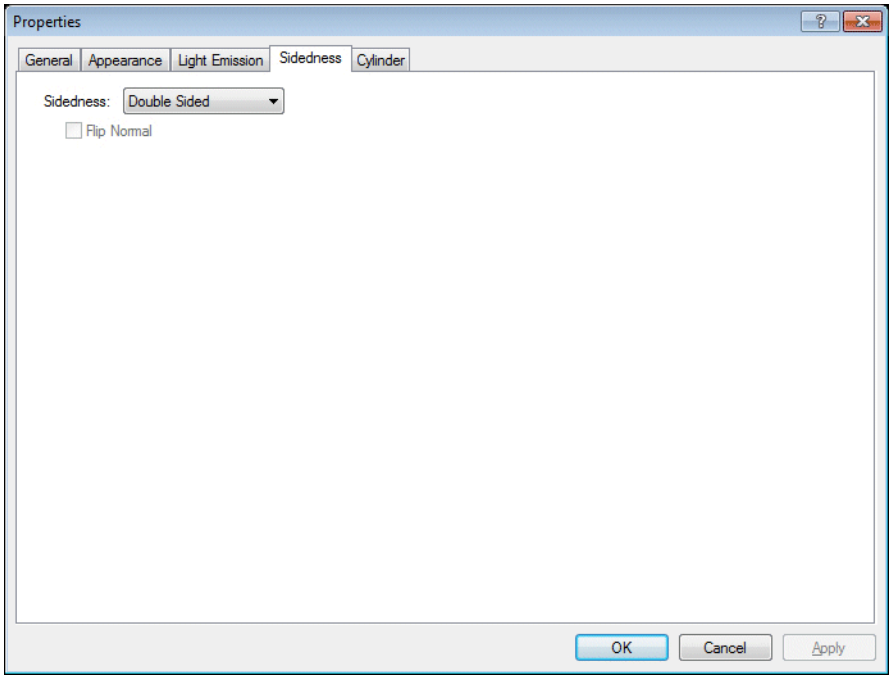
3 Click the *Sidedness* tab.

4 Click the appropriate option button, either *Double Sided* or *Single Sided*. To leave the single-sided object's faces oriented in the same direction in which they were drawn, proceed directly to step 6. To change the direction of the faces, see step 5.

5 If you are changing a double-sided object to single sided, and you want to change the direction in which the object's faces are oriented (either outward or inward), click the *Flip Normal* check box.

6 Click *OK*.

7 Review your changes in the *Shaded* tab and make adjustments as required.



- *Sidedness:* Choose the sidedness of the selected object:
  - *Double Sided*: Select this option to turn the single-sided object into a double-sided object. Note that you cannot change WYSIWYG's default venues from single to double sided.

- *Single Sided*: Select this option to turn the double-sided object into a single-sided object. For example, if you have created a custom surface/wall/venue and have imported it as a double-sided object into WYSIWYG, you can select it and make it single sided so you can see “into” the venue when you rotate it in Shaded views. Note that you cannot change any of WYSIWYG’s library objects from double to single sided.

- *Flip Normal*: If you have imported a custom-drawn object or drawn an object in WYSIWYG and you see in the Shaded view that it appears incorrectly (the “faces” showing the texture/color are pointing inward), select this check box to flip the faces in the opposite direction, and then look at the object in the Shaded view again.

Object-specific properties

As explained above, objects also have properties that are particular to the type of object. For example, fixtures have unit numbers but do not have a radius. When an object is selected and its properties are displayed, a tab appears in the properties dialog box for that type of object. When you select multiple objects of different types, tabs appear for each type of object selected. The following objects are defined:

- points
- lines
- text labels
- dimensions
- axes and motion frames

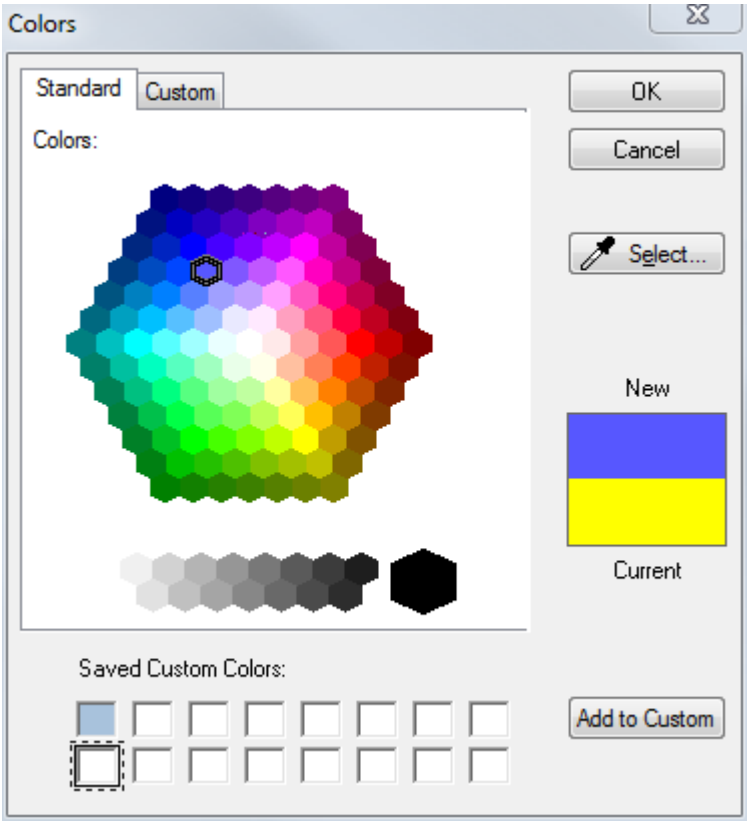
For all other types of objects (venues, circles, arcs, risers, cylinders, spheres, and cameras), refer to “Drawing objects” on page 138. In these cases, the properties dialog offers the same options that were given when the object was initially drawn.

Hanging structures properties (pipes, truss, floor mounts, and so on) are fully defined in “Hang structures” on page 257.

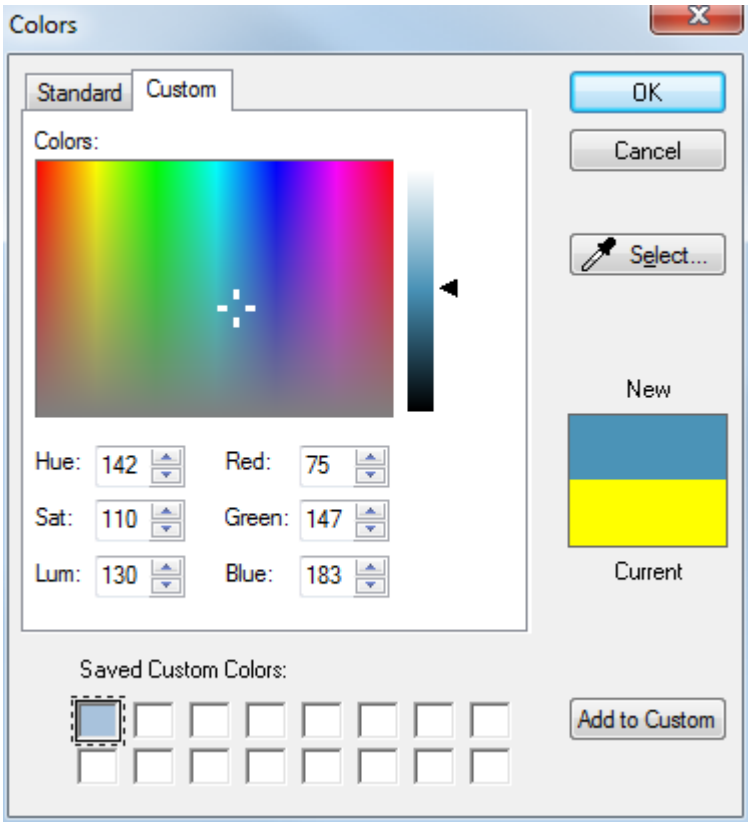
Fixtures and lighting-specific object properties are fully defined in “Hanging and focusing fixtures” on page 274.

Colors window

The Colors window allows for full customization and control over colors used in WYSIWG. From here the color of any object in WYSIWYG can be changed. This section will explain the various features of the color window.

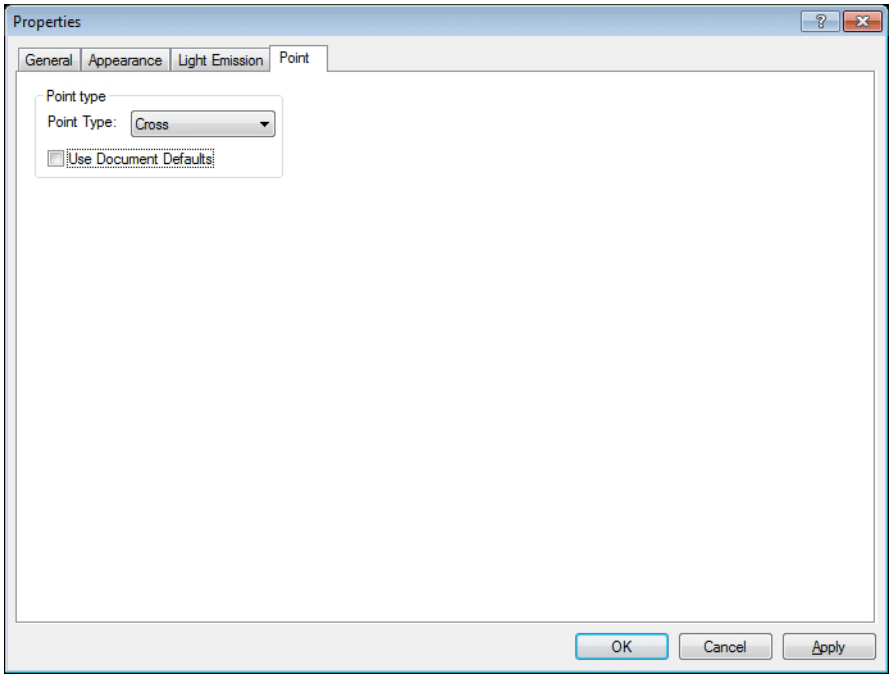


- *Standard*: A hexagon of basic colors and shades available for selection.
- *OK*: Will confirm the use of a new color to replace the current color.
- *Cancel*: Will exit the color window without making a color selection.
- *Select...*: This option is used to gather a custom color sample from anywhere on your desktop. This option will turn the cursor into a dropper from which colors can be gathered.
- *New*: Color selected in the Color window.
- *Current*: The color that is currently in use.
- *Saved Custom Colors*: A saved palette of custom colors.
- *Add to Custom*: Will save selected New color to the Saved Custom Colors palette.
- *Custom*: Section for editing a selected color's properties.



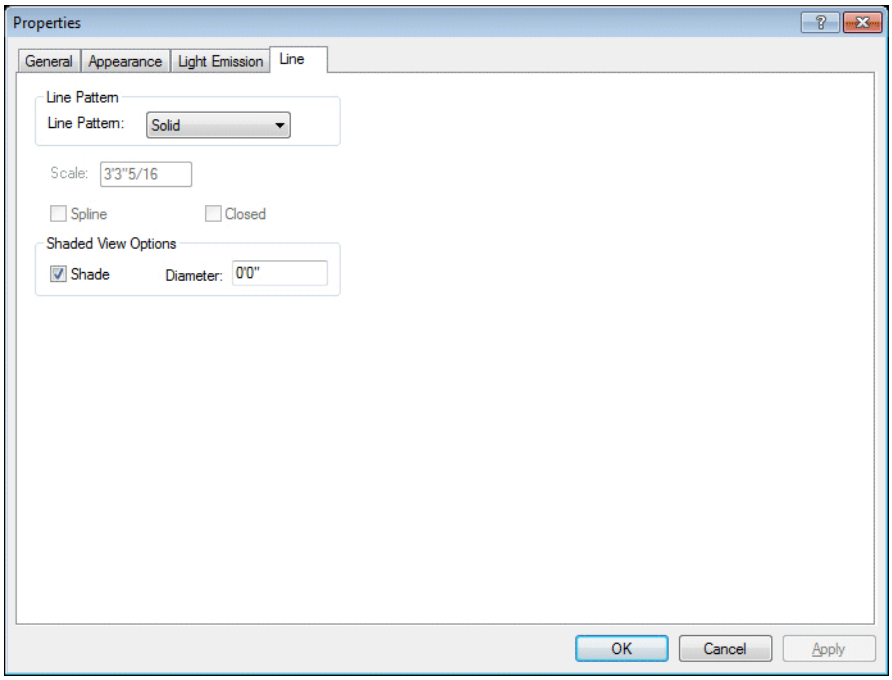
Point tab

Options on the *Point* tab affect how the selected point is drawn. The default point type used in a document is defined on the *Object Defaults* tab of *Show Options*. To ignore the default setting, click to clear the *Use Document Defaults* check box, and then select the desired point type.



Line tab

Options on the *Line* tab affect how the selected line is drawn.

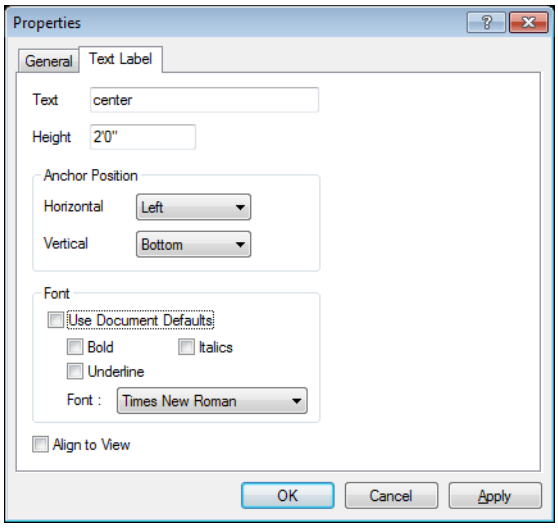


- *Line Pattern*: Choose a line pattern for the selected lines. Refer to “Drawing lines” on page 144 for an illustration of each type.
- *Scale*: Type a value in this box to change the length and spacing of dots and dashes for the selected lines. This value is applicable to center, hidden, or dot lines only.
- *Closed*: Select this check box to quickly connect the first point of a multi-segment line to the last point of that line.
- *Spline*: Select the *Spline* check box to transform a line into a spline or French curve.

  Note: You cannot change a line to a spline or French curve unless the selected line has more than two vertices.
- *Shade Options*: select the *Shade* check box to have the selected line(s) displayed in shaded views and renderings, and then type a value for the radius to set its thickness.

Text label tab

Options on the *Text Label* tab affect the justification and style of the selected text labels.

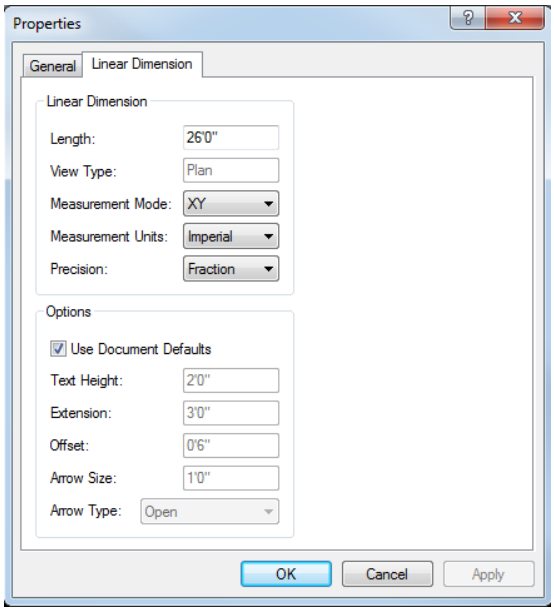


- *Text*: Type new text to change the label.

- *Height*: Type a value for the height to adjust the font size.
- *Horizontal Justification*: Select the Text label's horizontal alignment. The available horizontal alignment options are:
  - *Left*: Places the selected text label to the left of the insertion point.
  - *Center*: Horizontally centers the selected text label on the insertion point.
  - *Right*: Places the selected text label to the right of the insertion point.
- *Vertical Justification*: Select the text label's vertical alignment. The available vertical alignment options are:
  - *Top*: Places the selected text label below the insertion point.
  - *Center*: Vertically centers the selected text label on the insertion point.
  - *Bottom*: Places the selected text label above the insertion point.*Italics*: Select this check box to make the text appear in italics.
- *Font - Use Document Defaults*: Deselect this checkbox to choose the default font and font styles for all text labels, and then make your selections.
- *Align to View*: Select this check box to set the text labels to be legible regardless of plot type.

Linear Dimension tab

Options on the *Linear Dimension* tab affect the appearance and measurement mode of the selected dimension. To change the length of a dimension, it must be stretched in the drawing.

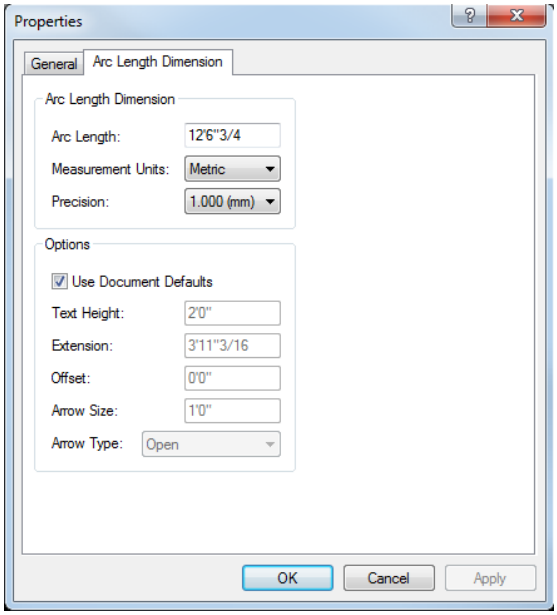


- *Length*: This box is not editable. It displays the actual length of the line drawn for the dimension.
- *View Type*: This box is not editable. It displays the plot type in which the dimension label is visible, which is determined when the dimension is drawn.
- *Measurement Mode:* Specify the view in which the dimension will be visible. Measurement modes are explained in "Drawing dimension objects" on page 150
- *Measurement Units*: Measurement modes are explained in "Drawing dimension objects" on page 150. Select *Imperial* or *Metric* to change the measurement units.
- *Precision*: Specify the measurement precision of the dimension line. Based on your choice for measurement units, you can choose to display the drawing resolution in centimeters, millimeters, whole numbers, or fractions.

- *Options - Use Document Defaults*: Toggle this option to specify whether the dimension is to use default document settings as configured in the Dimension Styles tab, or use custom settings.

Arc Length Dimension tab

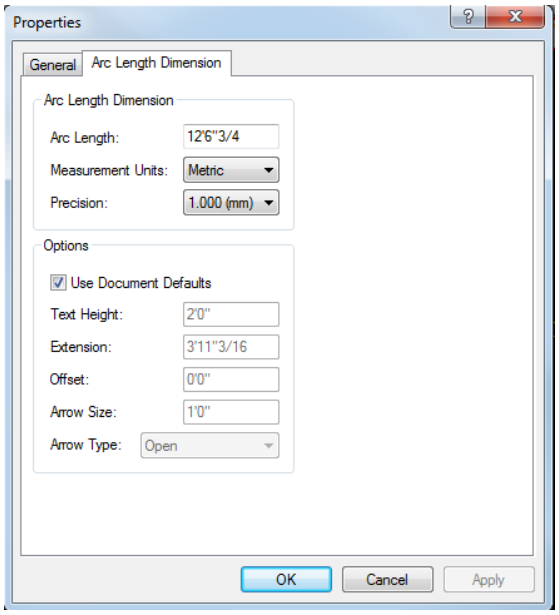
Options on the *Arc Length Dimension* tab affect the appearance and measurement mode of the selected arc length dimension. The arc length dimension can only be changed when the arc itself changes.



- *Arc Length*: This box is not editable. It displays the actual length of the arc to which the dimension is attached.
- *Measurement Units*: Measurement modes are explained in “Drawing dimension objects” on page 150. Select *Imperial* or *Metric* to change the measurement units.
- *Precision*: Specify the measurement precision of the dimension line. Based on your choice for measurement units, you can choose to display the drawing resolution in centimeters, millimeters, whole numbers, or fractions.
- *Options - Use Document Defaults*: Toggle this option to specify whether the dimension is to use default document settings as configured in the Dimension Styles tab, or use custom settings.

Radial Dimension tab

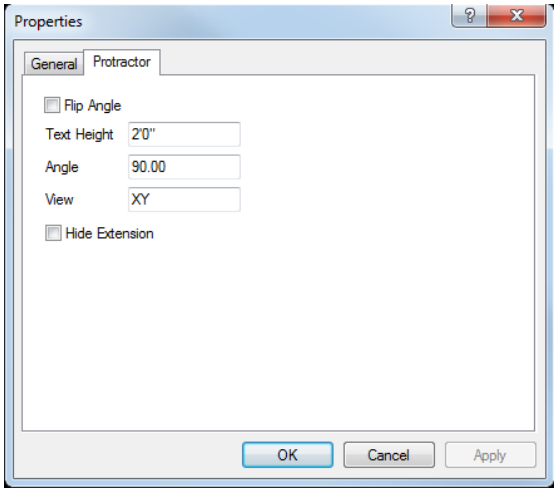
Options on the *Radial Dimension* tab affect the appearance and measurement mode of the selected radial dimension. The radial dimension can only be changed when the circle or arc itself changes.



- *Radius*: This box is not editable; it displays the actual radius of the circle or arc to which the dimension is attached.
- *Measurement Units*: Measurement modes are explained in “Drawing dimension objects” on page 150. Select *Imperial* or *Metric* to change the measurement units.
- *Precision*: Specify the measurement precision of the dimension line. Based on your choice for measurement units, you can choose to display the drawing resolution in centimeters, millimeters, whole numbers, or fractions.
- *Options - Use Document Defaults*: Toggle this option to specify whether the dimension is to use default document settings as configured in the Dimension Styles tab, or use custom settings.

Protractor tab

Options on the *Protractor* tab affect the appearance and measurement mode of the selected protractor object.

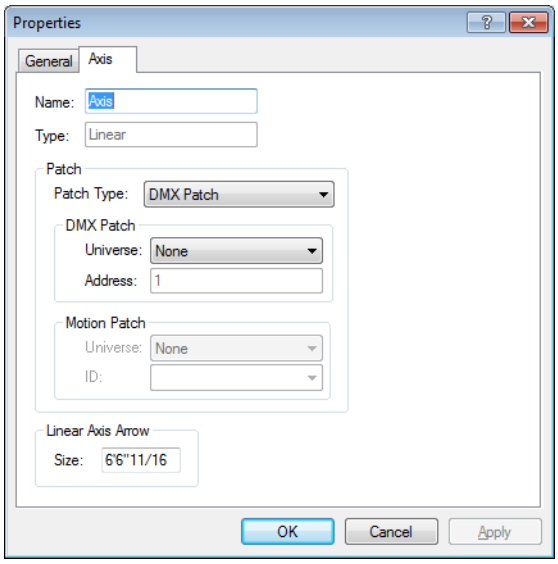


- *Flip Angle*: Toggle this option to switch between measuring the interior and exterior angle defined by the protractor tool.
- *Text Height*: Type a value to change the height of the text displayed by the protractor tool.
- *Angle*: The angle as measured by the protractor tool.

- *View*: The view that the angle was drawn in. Angles are only visible as long as they are drawn in a view coplanar to either XY, XZ, or YZ planes.
- *Hide Extension*: Toggle this option to hide or show the extension of the protractor object.

Axis tab

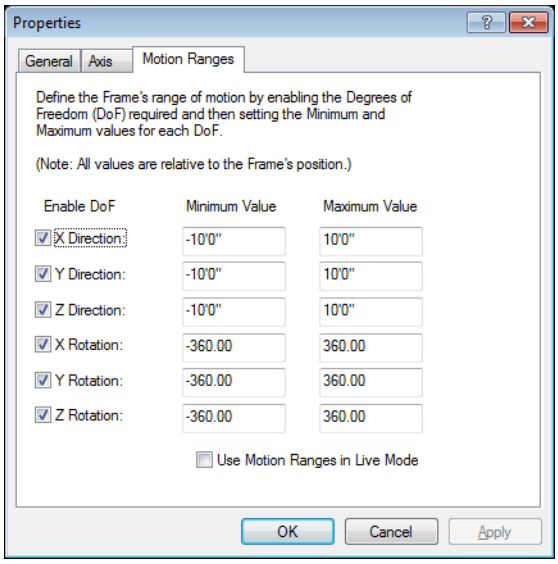
Options on the Axis tab affect the appearance of the selected axis or frame. The Axis tab also enables you to specify the patch settings for the selected axis or frame.



- *Name*: Modify this property to change the name of the axis.
- *Type*: Indicates the type of axis. Can be one of Linear, Rotation, or Frame.
- *Patch Type*: Allows you to specify which patch type you want. Can be one of DMX or Motion Patch. If *DMX Patch* is selected then the DMX Patch area will be enabled while the Motion Patch is disabled.
- *DMX Patch - Universe*: Select the DMX Universe to which you want to bind the axis. The Universe must be created in DATA mode.
- *DMX Patch - Address*: Specify the starting address of the axis object. Linear and Rotational axes take up two channels in a DMX universe (Move, Move Fine); Frames take 12 channels (X, X Fine, Y, Y Fine, Z, Z Fine, Rx, Rx Fine, Ry, Ry Fine, Rz, and Rz Fine).
- *Motion Patch – Universe*: Select the Motion Control Universe to which you want to bind the axis. The Universe must be created in DATA mode.
- *Motion Patch - ID*: Specify the root ID of the axis. Linear and Rotational axes will take up one slot in the Motion Control Universe; Frames will take up 6 slots in the Motion Control Universe, where each slot will take the form of ID:X, ID:Y, ID:Z, ID:Rx, ID:Ry, and ID:Rz.
- *Linear Axis Arrow*: Enter a value to change the size of the arrow on a selected linear axis. This option is not available if a rotation axis or a frame is selected.

Motion Ranges tab

Options in the Motion Ranges Tab enable you to modify the allowed ranges of motion for the selected Frame.



- *X Direction*: Enable this checkbox to allow linear movement along the X axis. Edit the minimum and maximum extents to define the Frame's motion range.
- *Y Direction*: Enable this checkbox to allow linear movement along the Y axis. Edit the minimum and maximum extents to define the Frame's motion range.
- *Z Direction*: Enable this checkbox to allow linear movement along the Z axis. Edit the minimum and maximum extents to define the Frame's motion range.
- *Rx Rotation*: Enable this checkbox to allow rotations around the X axis. Edit the minimum and maximum extents to define the Frame's motion range.
- *Ry Rotation*: Enable this checkbox to allow rotations around the Y axis. Edit the minimum and maximum extents to define the Frame's motion range.
- *Rz Rotation*: Enable this checkbox to allow rotations around the Z axis. Edit the minimum and maximum extents to define the Frame's motion range.
- *Use Motion Ranges in Live Mode*: When disabled, the specified motion ranges will only be enforced in DESIGN mode, giving you the full free range of motion in LIVE mode. When enabled, the specified ranges of motion will be enforced in LIVE mode as well.

Textures

You can apply textures to different objects, such as venues, surfaces, 3D primitives, and library objects. Surfaces are created using the *Surface* tool, or by extruding a line. The textures provided with WYSIWYG emulate various types of surfaces, such as brick or cloth. You can also use your own bitmap or jpeg files as textures, to emulate projection screens, or to cover a surface with a texture specific to your show.

When customizing an object, you can apply *either* a color or a texture; you cannot apply both.

To apply a texture to a surface

Note: The following procedure details how to apply a texture to a surface, but it can also be used to apply a texture to venues, risers, walls, library objects, spheres, cylinders, or cones by right-clicking, and then selecting **Properties**.

In order for textures to be displayed properly on surfaces, the surface must be drawn in a counter-clockwise direction. For details, see "To draw a surface" on page 153.

1. Right-click the surface to which you want to apply a texture, and then choose **Properties**.
2. Click the *Appearance* tab.
3. Select the texture option that you want. The options are:
   - *Use Texture from Library*: Indicates that the selected surface is a texture from the WYSIWYG library. The texture name appears in the box provided. Click the ellipsis button (...) to modify the selected surface.
   - *Use Texture from File*: The selected surface uses a texture from an external bitmap or jpeg file on your hard drive rather than one from the WYSIWYG library. Click the ellipsis button (...) to select the external file to be used as a texture.
4. Click *Tile* to have the texture repeated over the selected element in a continuous series of squares or rectangles, and then type the size of the frame in which you want the texture to appear in the *Width* and *Height* boxes. Based on the size that you enter, WYSIWYG calculates how many times the texture is repeated (or tiled) to completely cover the selected element.
5. Check *Stretch* to make the selected texture grow to cover the entire surface. Based on the aspect ratio and the rotation angle, WYSIWYG evaluates the surface with all of its edges and stretches the texture so the best fit is used. This option is not active unless you choose a texture. You cannot choose this option for spheres, cylinders, or cones. If you choose this option, you can also choose the following:
   - *Preserve Aspect Ratio*: Click this option to preserve the aspect ratio of the original texture image when it is stretched over the surface of the object you have selected. This option helps avoid distortion of complex textures. If the texture image cannot wrap completely over the entire surface of the object while maintaining its original aspect ratio, then the color that you choose in the *Border Color* box will be applied evenly around any excess surface area not covered by the texture (much like a picture frame around a picture).
   - *Border Color*: If you have chosen to preserve the texture's aspect ratio, click this box to choose the color that will be applied evenly around any excess surface area not covered by the texture (much like a picture frame around a picture)
6. If the texture has text or another recognizable image in it, you might need to rotate the image to get it right-side up. Select the rotation angle from the *Texture Rotation* drop-down box.
7. Click *OK*.

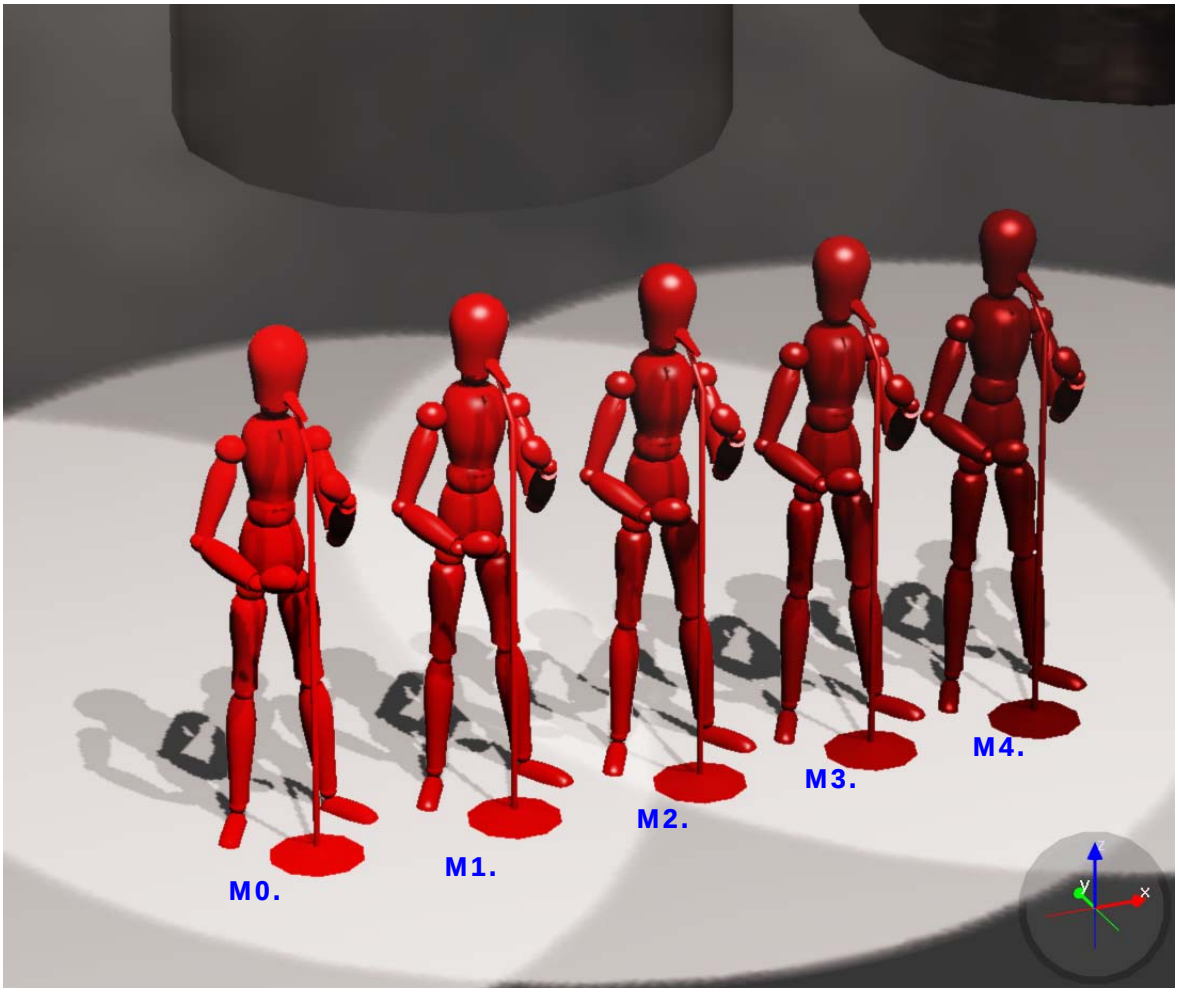
Materials

Materials differ from textures in that they do not affect what the objects or surfaces will look like, but rather how they interact with light. The result of applying a material is only apparent in renderings and in Shaded view. Below is an example of a rendering with materials included:



Objects and surfaces can be assigned both a texture and a material.

You apply materials using the *Appearance* tab in the *Properties* dialog box. You can choose your preferred material from the list of materials provided with WYSIWYG.



A sample of mannequins, exactly the same except for the material applied: M0 default settings, M1 Metal 1(Dull), M2 Metal 2, M3 Metal 3, M4 Metal 4 (Shiny).

To toggle materials in Shaded View

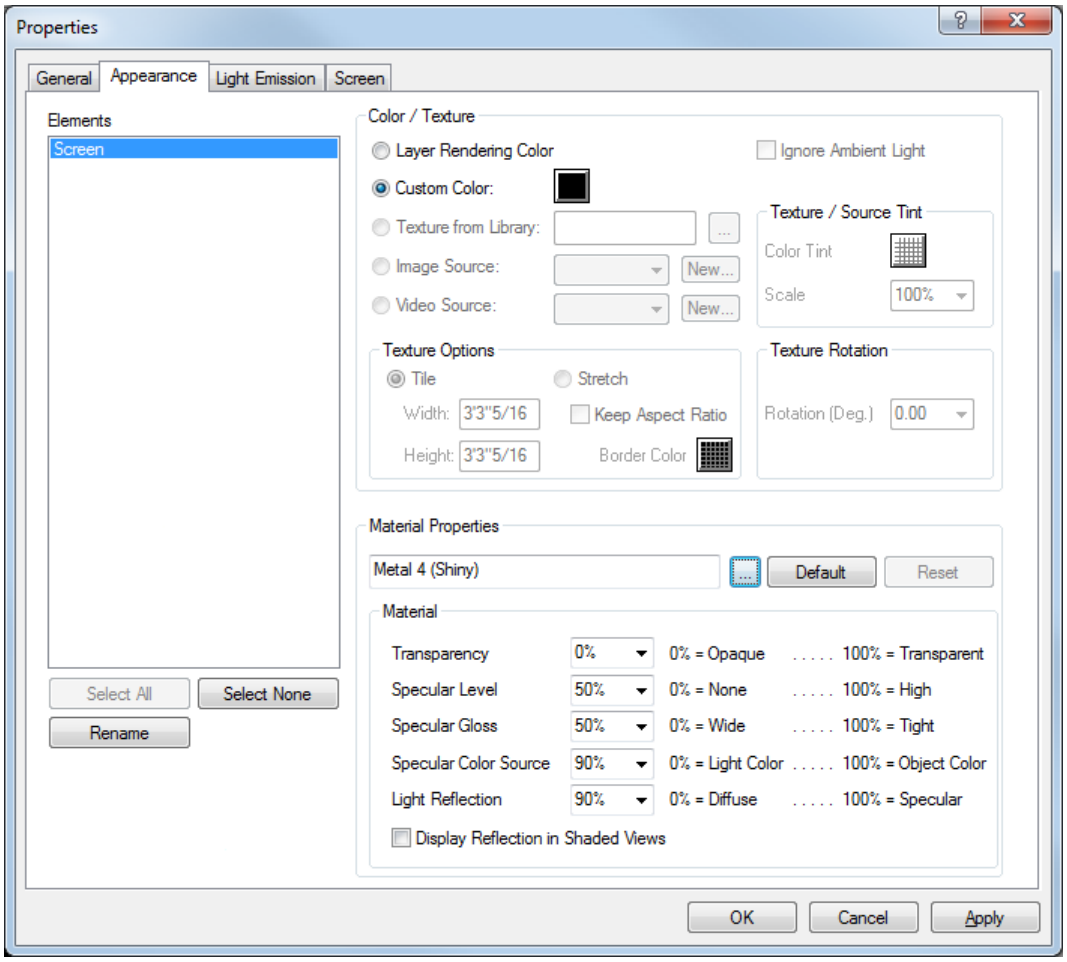
1 While in **Shaded** view, click **Options > View Options...**.

2 In the **Simulation** tab, click the **Materials** check box to toggle materials in shaded view on or off.

To apply a material to an object (surface, object, or venue)

1 Right-click the object which you want to apply the material to, and then choose **Properties**.

Result: The *Properties* window will appear.

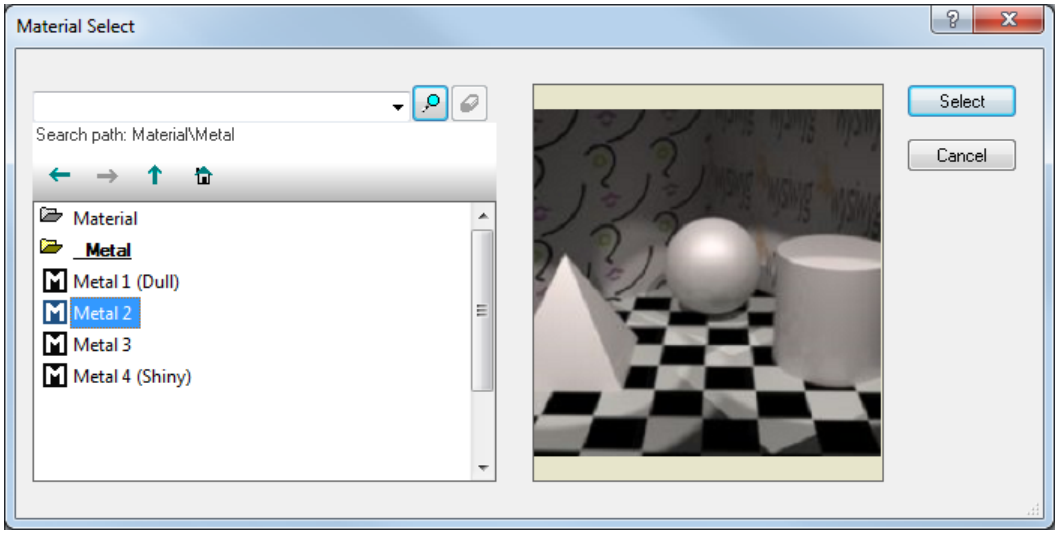


2 In the *Properties* window, click the *Appearance* tab.

3 From the **Elements** list box, choose the elements to which you want to apply the material. To choose a custom material for the selected elements (such as metal, paper, plastic, or fabric) click the ellipsis button (...) beside the *Material* box.

Result: The *Material Select* window will appear.

Tip: When a material is highlighted in the *Material Select* window, a preview of the material applied to objects is displayed.



4 In the *Material Select* window, navigate to and highlight the desired material, and then click *Select*.

Result: The settings of the element will be configured so that it simulates the material.

5 If desired, you can edit the default properties of the selected element. The element options vary as follows:

- *Transparency*: This setting controls the proportion of light that passes though the material.
- *Specular Level*: This setting controls how prominent other specular effects appear on the material.
- *Specular Gloss*: This settings controls the level of gloss a material will project.
- *Specular Color Source*: This setting controls the color of light projected on a material.
- *Light Reflection*: This setting controls the amount of light that is reflected off the material.

Note: Using the drop-down menus, you can change any of the material properties you as desired. To revert back to defaults, use the "*Reset*" button.

6 Click *Apply*.

7 Click *OK*.

Material Reflections

The reflective properties of a material can be toggled to enable true reflections in Shaded Views. When enabled, the material will produce a detailed and faithful representation of everything reflected on its surface.

Attention: The number of reflective material surfaces that WYSIWYG can support is **10**. When enabling reflections on an object, note the number of surfaces that make up the object. For example, a cube has 6 sides. If an entire cube was to be reflective, it would use 6 reflective surfaces.

Note: Objects that have reflections enabled will be displayed with a blue bounding box in the Shaded view



To apply reflections to an object (surface object or venue)

Note: The quality of reflection is based on the material properties of the object. Dull or unreflective materials will reflect poorly.

1 In **CAD** mode, **Wireframe** view, right-click the object which you want to apply the reflection to, and then choose **Properties**.

Result: The *Properties* window will appear.

2 In the *Properties* window, click the *Appearance* tab.

3 From the **Elements** list box, choose the elements of the object which you want to apply the reflection.

Note: Remember WYSIWYG limits the number of reflective surfaces to **10**.

4 In the **Material Properties** section, mark the checkbox by **Display Reflection in Shaded Views**.

5 Click *Apply*.

6 Click *OK*.

7 Go to **LIVE** mode, **Shaded** view.

8 Go to **Options > View Options...** .

Result: the *View Options* window will open.

9 In the *View Options* window, click on the *Simulation* tab.

10 Under the **Materials** section, mark the checkbox by **Reflections**.

11 Click *OK*.

Result: Reflective materials will be enabled and demonstrate true reflections.

Layers

Layers are drawing aids intended to help you organize your plot. They are like transparent acetate sheets upon which you can draw. In the same way that you can view several transparent sheets at once by placing them on top of each other, you can hide and unhide layers by choosing which sheets are in the stack. The top sheet is the layer that new objects are placed on and is called the current layer. You can also merge layers together if you decide that you need to combine one or more layers.

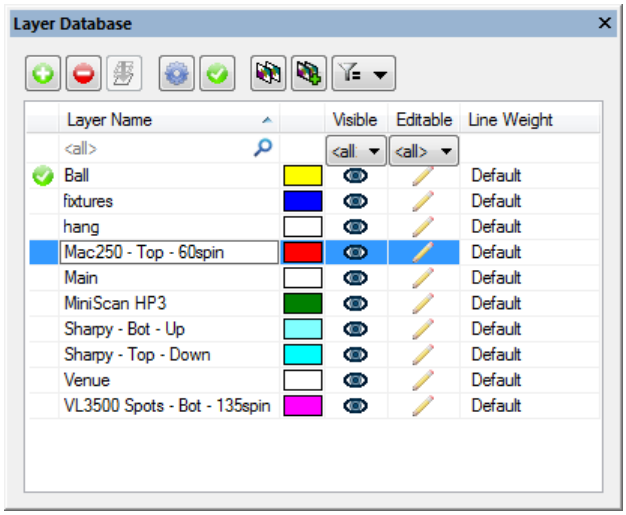
To find out the layer to which any object in your plot is assigned, just hover over the object with your cursor in any of the wireframe views. A tooltip appears listing the object name and its layer. This is especially useful when you are working with multiple layers, some of which have the same color.

Creating layers

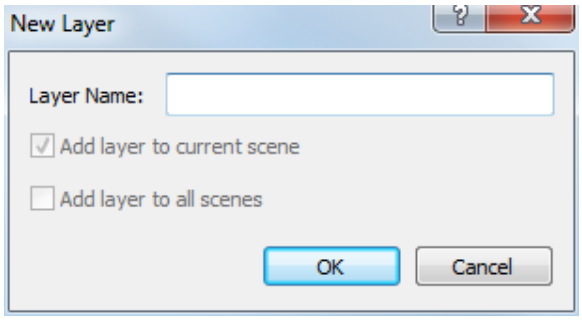
To create new layers

1 From the **Edit** menu, choose **Layers**.

Result: The *Layer Database* dialog box is displayed.



2 To create a new layer, click the *New* icon. 



3 Type a new name for your layer (for example, “Stage”).

4 Choose whether to add the layer to the current Scene only or to all existing Scenes. (WYSIWYG maintains this setting for the next layer that you create.)

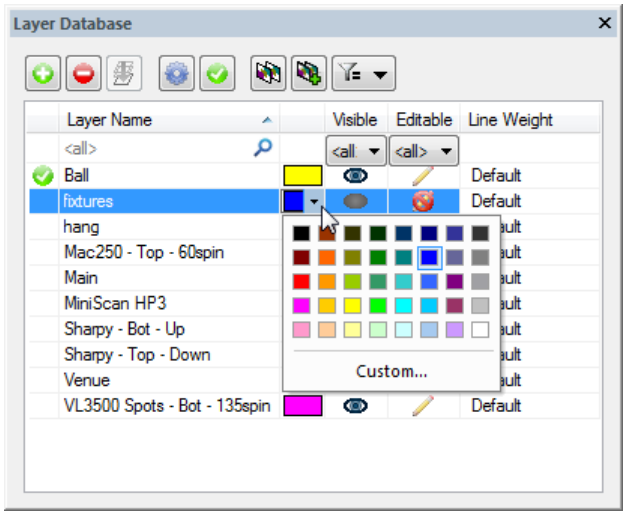
5 Click *OK*.

6 To make a layer and all objects on the layer visible on the plot and in shaded views, select the *Visible* icon. If not selected, the layer is not visible and is not, therefore, editable.

7 To make a layer and all objects on the layer editable on the plot and in shaded views, select the *Editable* icon. If not selected, the layer is not editable and is not, therefore, visible.

Tip: You might want to deselect a layer when you are finished working with it to avoid possible errors while working on other parts of the plot. This is similar to freezing in Auto CAD.

8 To change the color of the layer, click *Color Select*.



Note: It is recommended that you select different colors for each layer so as to easily identify the layers on the plot.

9 To change the line weight of the selected layer, choose the applicable thickness from the drop-down box. If you leave *Default* selected, the line weight from the Show Options tab is applied to the selected layer. The line weight that you select applies to all objects on this layer (including library objects, pipes, and truss) and is visible in all wireframe views in all modes.

10 To view the properties of a layer (for example, name, color, and inventory of objects drawn on the layer), click the Layer Properties icon. For more information on these properties, refer to “Layer properties” on page 220.

11 Before closing the dialog, set your current layer by highlighting it, and then clicking *Set Current*. . Alternately, you can set the current layer buy double clicking in the first column on the left, beside a layer name.

Note: The current layer is the layer that you are working on at the moment – any object that you draw is placed on this layer and assumes the layer's default properties when it is drawn. A check mark appears beside the name of the current layer.

12 To view and modify the scenes in which a layer is included, select a layer, and then click *Scenes*. For details, see “Scenes” on page 223.

13 Click *OK* to close the Layers dialog.

Note: To quickly sort the layers within the layer database, click the appropriate column heading in the *Layer Database* dialog box. For example, click the *Editable* column heading to sort the layers by edit setting; that is, those layers marked as editable appear at the top of the list in alphabetical order.

Merging layers

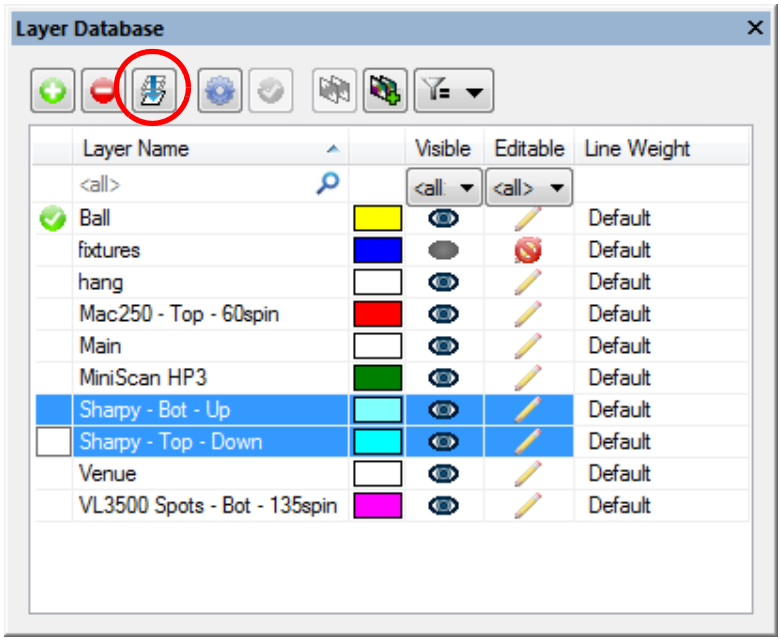
After you add layers, you can merge two or more of them together if you decide that you need to combine them.

To merge layers

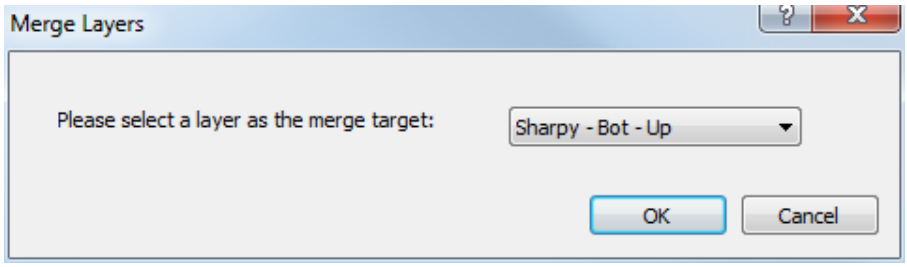
1 From the **Edit** menu, choose **Layer...**.

2 In the Layer Database window, hold CTRL and click to select the layers that you would like to merge together.

3 With the layers selected, click the *Merge Layers* button  .



Result: WYSIWYG asks you to specify the Target Layer, or the layer to which the additional layers will be merged.



4 From the drop-down box, select the target layer, and then click *OK*.

Result: The layers are merged, and you will see only the target layer (the one you chose to merge other layers into).

Layer properties

All layers have properties that describe the appearance of the layer. Objects that reside on the layer inherit the properties of the layer by default unless you decide to override the values. For more information on overriding object properties, refer to “General object properties” on page 195.

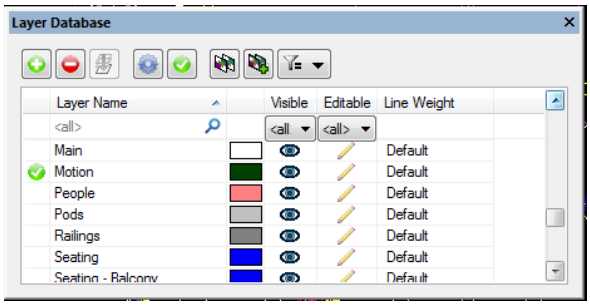
Adding filters to the Layer Database

When using scenes, the Layer Datebase can have filters applied so that only layers found in the scene will be visible.

To filter the Layer Database

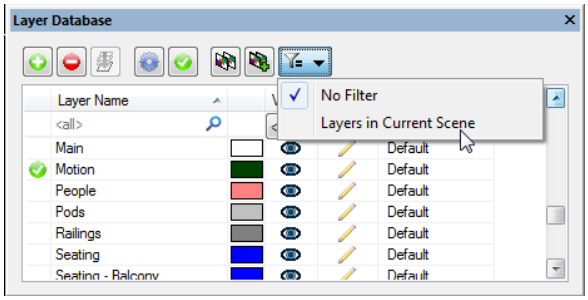
1 From the **Edit** menu, choose **Layers**.

Result: The *Layer Database* dialog box is displayed.



2 In the Layer Database window, click the *Filter* icon. 

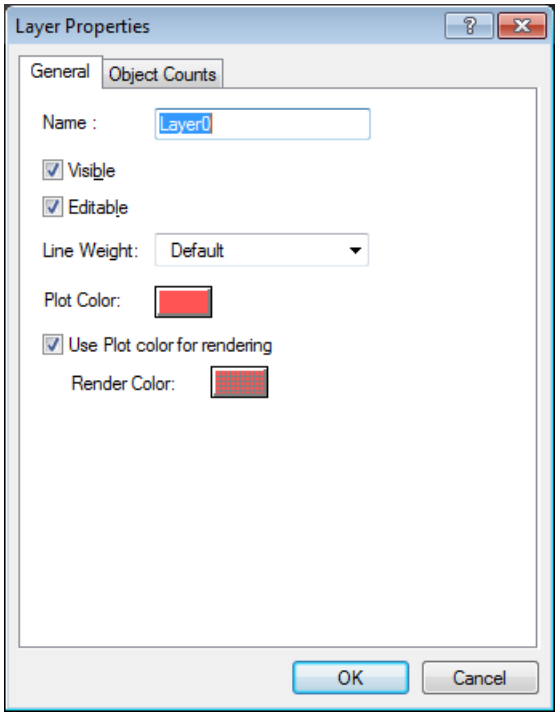
3 In the filter list, select **Layers in the Current Scene.**



Result: Only layers in the current Scene will be visible in the Layer Database

General tab

Options on the *General* tab affect the selected layer's color, line weight, and visibility.

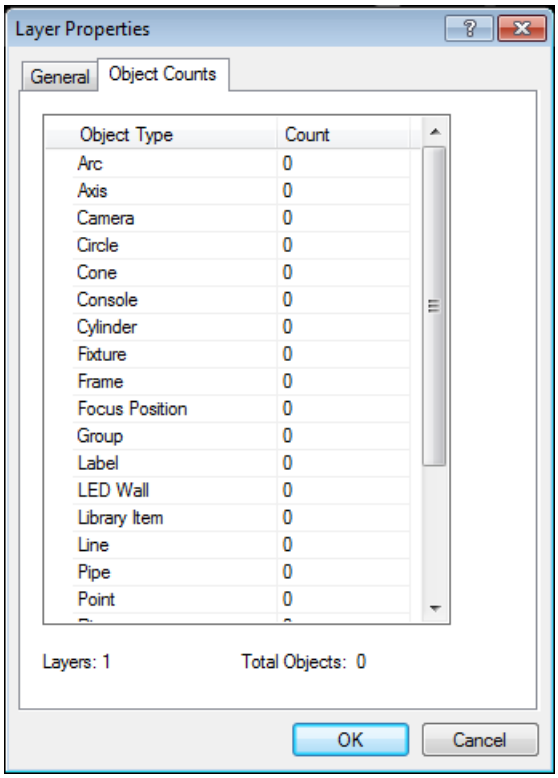


- *Visible*: Select this check box to make the selected layer visible on the plot and in shaded views. If you modify the property here, it is changed on the Layer Database dialog box.
- *Editable*: select this check box to make the selected layer editable on the plot and in shaded views. Modifying the property here changes it on the Layer Database dialog box.
- *Line Weight*: From the drop-down box, choose *Default* to set the layer's line weight to the default value specified in Show Options. For more information, refer to the "Show Options window" on page 395.

  If preferred, you can also choose a specific line weight for the layer. The value is used as the line weight for all the objects that reside on the specified layer (including library objects, pipes, and truss), unless otherwise modified. To modify the line weight of an object, refer to the "General tab" on page 195.
- *Plot Color*: Sets the layer's color for wireframe views. This is the color that is used for objects that reside on the selected layer when printing the document, unless otherwise modified. To modify the object color, refer to the "General tab" on page 195.
- *Use Plot Color for rendering*: select this check box to set the layer's color for shaded views and renderings to the same as the plot color specified above. Click to clear the check box and use the color button to set a specific color for the selected layer. The chosen color is used for all objects that reside on the selected layer, unless otherwise modified. To modify the object color for rendering, refer to the "General tab" on page 195.

Object Counts tab

Entries on the *Object Counts* tab indicate the total number of objects that reside on the selected layer. Values on this tab are read-only.



Deleting layers

You must remove all objects from a layer before you can delete it. You cannot delete the current layer.

To delete a layer

1. Select all the objects on the layer that you want to delete, and then delete them or move them to another layer.
2. From the **Edit** menu, choose **Layers**.

   Result: The *Layer Database* dialog box appears.
3. Ensure that the layer you want to delete is not set as the current layer.
4. Select the layer from the list.
5. Click the Delete Layer(s) icon. 
6. Click *OK*.

Tips:

- The *Layer* toolbar gives you quick access to all existing layers and the Layers database. Click the tool on the left end of the *Layer* toolbar to access the Layer database. 
- Use the layer list in the *Layer* toolbar to quickly set the current layer. If objects are selected when you do this, those objects are moved to the selected current layer.

Scenes

Scenes are collections of layers used to organize the plot. A scene may contain one or more layers.

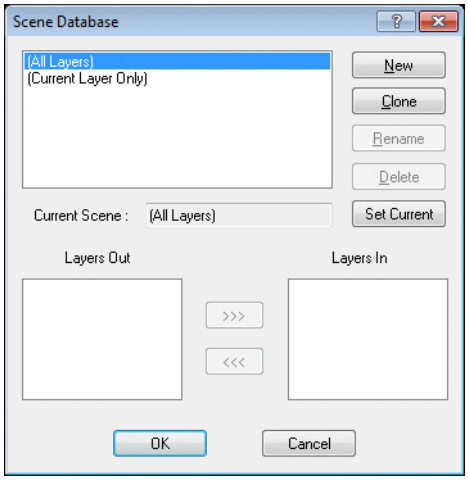
Hiding a scene hides all the layers assigned to that scene. By organizing the layers and scenes in your plot, you can quickly perform set changes.

By default WYSIWYG creates two initial scenes: *Current Layer Only* and *All Layers*. You cannot delete these scenes.

To create a new scene

1 From the **Edit** menu, choose **Scenes**.

Result: The *Scene Database* dialog box is displayed.



2 Click *New* to create a new scene.

3 Type the name of the new scene, and then click *OK*.

Result: The layers associated with the scene are displayed in the *Layers Out* and *Layers In* boxes.

4 Select the layers that you want to include in the scene, and then use the arrow buttons to move the layers accordingly. The layers that are available in a scene can also be set in the *Layers* dialog box. For more information, refer to “Layers” on page 218.

Notes:

- To copy a scene, select the scene, and then click *Clone*. The copied scene has the same properties of the selected scene.
- To rename a scene, select the scene, and then click *Rename*.
- To remove a scene from the scene database, select the scene, and then click *Delete*.

5 Click *Set Current* to activate your new scene.

Note: If the current layer is not in the current scene, you will not be able to add any objects.

6 Click *OK*.

Tip:

The *Scene* toolbar gives you quick access to all existing scenes and the Scenes database. Click the tool on the left end of the *Scene* toolbar to access the Scene database. 

Note: In order for fixtures to be displayed in scenes, both the layer on which the fixtures reside and the layer on which the respective hang structure resides (if different) must be included in the scene. You cannot display fixtures without their hang structures.

CAD tools

Break

The Break command allows you to break an object into smaller components.

To break an arc into arcs

1 Select the arc that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Arc into arcs**.

3 Type the number of desired break arcs, and then click *OK*.

Result: The arc is broken equally into the number of arcs specified.

To break an arc into lines

1 Select the arc that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Arc into lines**.

3 Type the number of desired break lines, and then click *OK*.

Result: The arc is broken equally into the number of lines specified.

To break a circle into arcs

Note: You can break circles into arcs only if they are perfectly round. If you have created a circle with a different horizontal and vertical radius, then you cannot break the circle into arcs.

1 Select the circle that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Circles into arcs**.

3 Type the number of desired break arcs, and then click *OK*.

Result: The circle is broken equally into the number of arcs specified.

To break a circle into lines

1 Select the circle that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Rectangle at Vertices**.

Result: The rectangle is broken into lines at each vertices.

To break a rectangle at vertices

1 Select the rectangle that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Circles into lines**.

3 Type the number of desired break lines, and then click *OK*.

4 The circle is broken equally into the number of lines specified.

To break a line at vertices

1 Select the multi-segment line that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Lines At Vertices**.

Result: The continuous multi-segment line is broken at the segment intersections. Each vertex is now an independent line.

To break a line into lines

1 Select the line or multi-segment line that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Lines into lines**.

3 Type the number of desired break lines, and then click *OK*.

Result: The line is broken equally into the number of lines specified.

To break a solid into surfaces

You can use this procedure to break solids like library objects, cones, and risers down into surfaces that you can customize individually. Note that when you do so, all of the object's original appearance elements and sidedness properties remain intact.

1 Select the 3D solid that you want to break, a riser for example.

2 From the **Tools** menu, choose **Break**, and then choose **Solids into Surfaces**.

Result: Each segment of the solid becomes an independent surface.

Note: Alternatively, to perform a break command, you can use the *Break* tool on the *Tools* toolbar.  This is the same as choosing **Specify** from the **Break** sub-menu.

To break a solid into lines

You can use this procedure to break solids like library objects, cones, and risers down into lines. Note that when you do so, all of the object's original appearance elements and sidedness properties remain intact.

1 Select the 3D solid that you want to break, a riser for example.

2 From the **Tools** menu, choose **Break**, and then choose **Solids into Lines**.

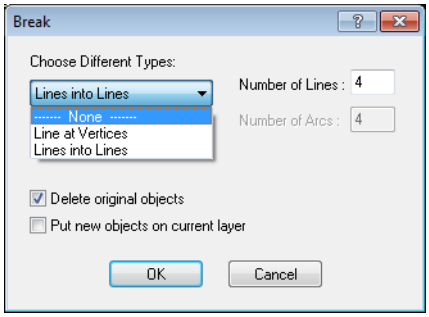
Result: Each segment of the solid becomes an independent line.

Note: Alternatively, to perform a break command, you can use the *Break* tool on the *Tools* toolbar.  This is the same as choosing **Specify** from the **Break** sub-menu.

To break using the Specify dialog

1 Select the object that you want to break.

2 From the **Tools** menu, choose **Break**, and then choose **Specify**.



3 Select the desired break type from the available options by selecting the appropriate check boxes. Based on the object that you have selected to break, these options vary.

4 In the boxes at the top right corner, specify the number of break lines or arcs if appropriate.

5 Select the *Delete original objects* check box if you want the original object to be replaced with the new “broken” ones.

6 Select the *Put new objects on current layer* check box if you want the new “broken” objects to be placed on the current layer, as opposed to the selected object’s original layer, if different.

7 Click *OK*.

Convert

The Convert command turns an object into a different kind of object. For example, you can convert a circle into a circular surface, you can convert lines into pipes or surfaces, and you can convert points into focus positions, and vice versa.

To convert a circle into an arc

Note: You can convert circles into arcs only if they are perfectly round. If you have created a circle with a different horizontal and vertical radius, then you cannot convert the circle into an arc.

1 Select the circle that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Circles into arcs**.

Result: The circle is converted into a 360-degree arc. The marker points can be dragged out to open the arc, if desired.

To convert a circle into a surface

1 Select the circle that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Circles into Surfaces**.

3 Type the number of segments to make up the contour of the surface. You cannot type the number 1. The higher the number, the more circular the surface.

Result: The circle is converted into a surface.

Notes:

- If you type 2, the circle is converted into a linear surface.

- If you type 3, the circle is converted into a triangular surface.
- If you type 4, the circle is converted to a square surface.
- This pattern will repeat for each increment in the number of segments.

To convert a line into a surface

You can use this tool to convert a multi-segment Line object into a surface. Note that if the Line is not closed, this operation will close it by connecting the start and end points.

1 Select the Line object that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Lines into Surfaces**.

Result: The line is converted into a surface.

To convert a line into a pipe

1 Select the line that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Lines into pipes**.

Result: The line is converted into a pipe, assuming all of the properties and characteristics of a pipe. You can name it and hang fixtures on it.

To convert a pipe into a line

1 Select the pipe you want to convert.

Note: There cannot be any fixtures hanging on the selected pipe. You must unhang or delete any fixtures before attempting to convert the pipe into a line.

2 From the **Tools** menu, choose **Convert**, and then choose **Pipes into lines**.

Result: The pipe is converted into a line.

To convert a focus position into a point

1 Select the focus position that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Focus positions into points**.

Result: The focus position is converted into a point. Any fixtures that were focused to the initial focus position retain their orientation, but lose the association with that point.

To convert a point into a focus position

1 Select the point that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Points into focus positions**.

Result: The points are converted into a focus position, and are automatically called “Label.” To change the name for the newly created focus position, refer to “Focus position properties” on page 282.

Note: Alternatively, to perform a convert command, you can use the *Convert* tool on the *Tools* toolbar. 

To convert a line into a focus line

1 Select the line object that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Lines into Focus Lines**.

Result: The line object is converted into a focus line.

To convert a focus line into a line

1 Select the focus line that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Focus Lines into Lines.**

Result: The line object is converted into a focus line. Any fixtures that were focused to the initial focus position retain their orientation, but lose the association with that line.

To convert an arc into a focus arc

1 Select the arc object that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Arcs into Focus Arcs.**

Result: The arc object is converted into a focus arc.

To convert a focus arc into an arc

1 Select the focus arc that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Rectangles into Lines.**

Result: The arc object is converted into a focus arc. Any fixtures that were focused to the initial focus position retain their orientation, but lose the association with that arc.

To convert a rectangle into pipes

1 Select the focus rectangle that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Rectangles into Pipes.**

Result: The rectangle object is converted into lines.

To convert a rectangle into surfaces

1 Select the focus rectangle that you want to convert.

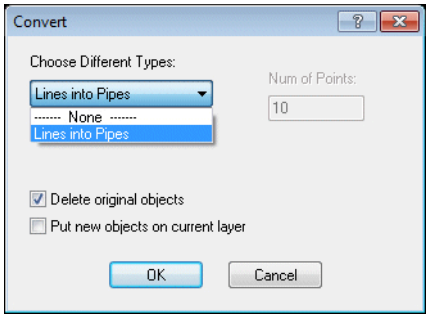
2 From the **Tools** menu, choose **Convert**, and then choose **Rectangles into Surfaces.**

Result: The rectangle object is converted into a surface.

To convert using the Specify dialog

1 Select the object that you want to convert.

2 From the **Tools** menu, choose **Convert**, and then choose **Specify**.



3 Select the desired conversion type from the available options by selecting the appropriate option buttons. Select the *Don't convert* option button if you do not want to convert the circle into either an arc or a surface.

Note: For circles, select the *Arcs* or S*urfaces* option button, and then type the number of points as discussed above.

4 Select the *Delete original objects* check box if you want the original object to be replaced with the new "converted" one.

5 Select the *Put new objects on current layer* check box if you want the new "converted" objects to be placed on the current layer, as opposed to the selected object's original layer, if different.

6 Click *OK*.

Divide

The divide command separates objects into equal parts. The initial object is left intact (if not deleted), and the parts are identified by newly created objects, such as points or lines.

To divide an arc into lines

1 Select the arc that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Arcs into lines**.

3 Type the number of lines desired.

Result: The arc is divided into equal parts by the number of lines specified. The lines are automatically drawn from the center of the arc outward, resulting in something similar to a pie chart.

To divide an arc into points

1 Select the arc that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Arcs into points**.

3 Type the number of points desired.

Result: The arc is divided into equal parts by the number of points specified. The points are placed along the arc at the calculated interval.

To divide a circle into lines

1 Select the circle that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Circles into lines**.

3 Type the number of lines desired.

Result: The circle is divided into equal parts by the number of lines specified. The lines are automatically drawn from the center of the circle outward, resulting in something similar to a pie chart.

To divide a circle into points

1 Select the circle that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Circles into points**.

3 Type the number of points desired.

Result: The circle is divided into equal parts by the number of points specified. The points are placed along the circle at the calculated interval.

To divide a cylinder into circles

1 Select the cylinder that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Cylinders into circles**.

3 Type the number of circles desired.

Result: The cylinder is divided into equal parts by the number of circles specified. The circles are stacked within the cylinder at the calculated interval.

To divide a cylinder into lines

1 Select the cylinder that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Cylinder into lines**.

3 Type the number of lines desired.

Result: The cylinder is divided into equal parts by the number of lines specified. The lines follow the same pattern as the cylinder's existing segments, but at the calculated interval.

To divide a line into points

1 Select the line that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Lines into points**.

3 Type the number of points desired.

Result: The line is divided into equal parts by the number of points specified. The points are placed along the line at the calculated interval.

Note: Lines may also be divided by adding vertices. In this case, no new objects are created; the line is simply divided by marker points.

To divide a rectangle into points

1 Select the rectangle that you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Rectangle into points**.

3 Type the number of points desired.

Result: The rectangle is divided into equal parts by the number of points specified. The points are placed along the rectangle perimeter at the calculated interval.

To add a vertex to a line

1 Select the line to be divided.

2 Right-click on the line segment that you want to divide, and then choose **Add Vertex**.

Result: The original line remains intact. A marker point is placed at the midpoint of the line.

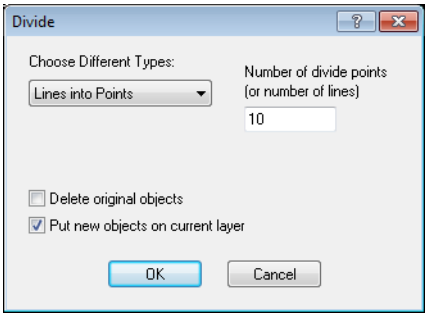
Notes:

- This procedure can be repeated for vertices within a line. The marker points will always be placed at the mid point of the vertices.
- Marker points can be dragged out to re-shape the line.
- If the line is deleted, the marker points are also deleted.
- Alternatively, to perform a divide command, you can use the *Divide* tool on the *Tools* toolbar.  This is the same as choosing *Specify* from the *Divide* sub-menu.

To divide using the Specify dialog

1 Select the object you want to divide.

2 From the **Tools** menu, choose **Divide**, and then choose **Specify**.



3 Select the desired divide type from the available options by selecting the appropriate check boxes.

4 Select the *Delete original objects* check box if you want the original object to be deleted when the new ones are created.

5 Select the *Put new objects on current layer* check box if you want the new objects to be placed on the current layer, as opposed to the selected object's original layer if different.

6 Click *OK*.

Extrude

The Extrude command allows you to turn a 2D object like a line or an arc into a surface, or turn a surface into an extruded surface or 3D solid.

To extrude a line

1 Select the line that you want to extrude.

2 From the **Tools** menu, choose **Extrude**, and then choose **Lines into Surfaces**.

3 Type the distance to extrude the line. The extrusion will occur parallel to the axis of the missing coordinate. For example, a line shown in plan view will extrude vertically. For splines, you must specify the number of segments for the new surface in the *# of Points* box.

To extrude an arc

1 Select the arc that you want to extrude.

2 From the **Tools** menu, choose **Extrude**, and then choose **Arcs into Surfaces**.

3 Type the distance to extrude the line. The extrusion will occur parallel to the axis of the missing coordinate. For example, an arc shown in plan view will extrude vertically.

4 Type the number of segments for the new surface. You cannot type the number 1. The higher the number, the more circular the surface.

To extrude a surface into an extruded surface

1 Select the surface that you want to extrude.

2 From the **Tools** menu, choose **Extrude**, and then choose **Surface into Extruded Surface**.

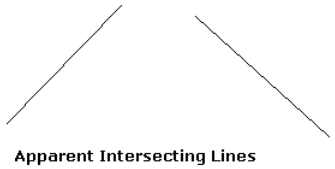
3 Type the distance to extrude the surface. The extrusion occurs parallel to the axis of the missing coordinate. For example, a surface shown in plan view will extrude vertically.

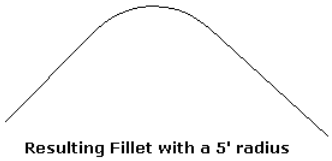
Tip:

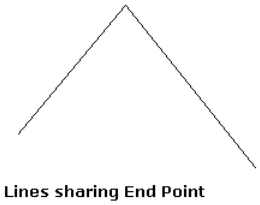
You can also use the *Extrude* tool on the *Tools* toolbar. 

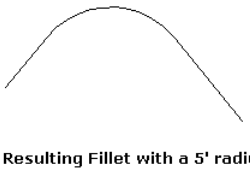
Fillet

The Fillet command joins two lines sharing the same end point or two apparent intersecting lines with an arc. A fillet radius must be specified to achieve the desired fillet angle.









To set the fillet radius

1 From the **Tools** menu, choose **Fillet**, and then choose **Specify fillet radius**.

2 Type a radius measurement for the resulting fillet.

Note: All subsequent fillet commands will follow this value until it is modified again.

To fillet two lines

1 Select the lines to be filleted.

2 From the **Tools** menu, choose **Fillet**, and then choose **Fillet**.

Result: The lines are joined by an arc based on the value set for the fillet radius.

Notes:

1. You must set a fillet radius before you can perform this command.
2. If the fillet radius is 0, the lines will be extended and joined at their intersection. If they are already intersecting, nothing occurs.

Align

You can use the Align tools to quickly and precisely align a series of objects that you have drawn or have inserted from the Library into your drawing. For example, if you have manually placed some chairs in your drawing, you can select them all and align them in one of six ways.

Notes: You cannot use the Align tool to align fixtures.

You can align the selected objects from the:

- top
- bottom
- left
- right
- center (horizontally)
- center (vertically)

Before choosing any of these options, you must select one of following three options:

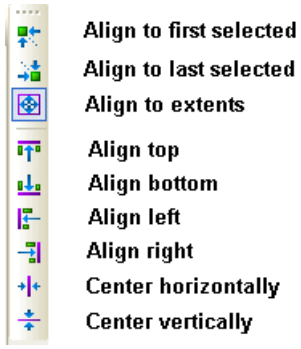
- **To First Selected**: Objects selected after the first object was selected will align (Left, Right, Centered Horizontally, etc.) to that first object.
- **To Last Selected**: Objects selected before the last object was selected will align (Left, Right, Centered Horizontally, etc.) to that last object.
- **To Extents**: All selected objects (regardless of selection order) will align to the left-most, right-most, bottom-most, etc. point of the left-most, right-most, bottom-most, etc. object, respectively.

To align objects

Note: You can align objects using the Align tools, or the CAD Align toolbar. The following procedure details how to use the CAD Align toolbar.

1 Select the objects that you want to align by clicking and dragging a selection box around them, or selecting them individually.

2 Ensure that the CAD Align toolbar is visible.



3 Choose your preferred alignment option:

- To First Selected
- To Last Selected
- To Extents

Note: Before you align the objects, it is recommended you first verify that they will not overlap each other in their new positions.

4 On the toolbar, select how you would like to align the selected objects:

- Top
- Bottom
- Left
- Right

- Center Horizontally
- Center Vertically

Array

The Array command is used to make multiple copies of an object distributed evenly along a line, arc, in a grid pattern, or, in the case of fixtures, along a pipe or straight length of truss.

Note: For details about array fixtures along a pipe or truss, see “Hanging multiple fixtures” on page 275.

To perform a linear array

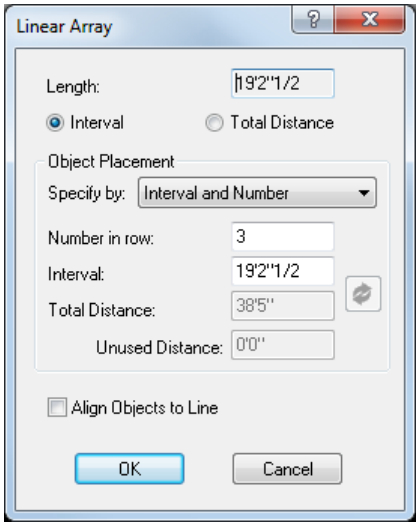
1 Select the object that you want to copy.

2 From the **Tools** menu, choose **Array**, and then choose **Array Linear**.

3 Click a point on your drawing or type in coordinates to specify the direction and interval for the linear array. For example, to place a copy of the selected object every 5’ at a 45-degree angle, type the following, and then press ENTER:

@5’<45

Result: The *Linear Array* window appears.



4 In the *Linear Array* window, in the **Length** section, select whether the section drawn in **Step 3** will be an *Interval*, or the *Total Distance* of the array.

5 In the **Object Placement** section, select which values will be available for editing.

Note: When performing an array copy, the value not chosen for editing is calculated automatically by WYSIWYG and is grayed out.

Note: To undo changes made to the copy values, click *Reset* button  .

6 Enter the total number of copies desired (this includes the original object).

7 Enter the interval distance between objects.

8 Enter the distance from the first to last object.

9 Select whether to align objects to the line.

10 Click *OK*.

To perform a array along a path

Perform this procedure to copy a selected object along a path that shares the same design as a line, arc or spline.

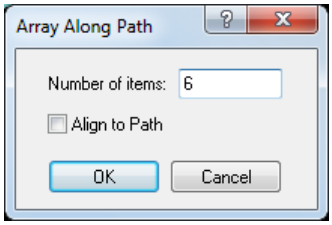
1 Draw a line, arc or spline that will act as a path for the cloned object.

2 Select the object you want to copy.

Note: The object will be copied from its current location. To copy the object on top of the path, snap the object to the end point of the path.

3 From the **Tools** menu, choose **Array**, then choose **Array Along Path**.

Result: The *Array Along Path* window appears.

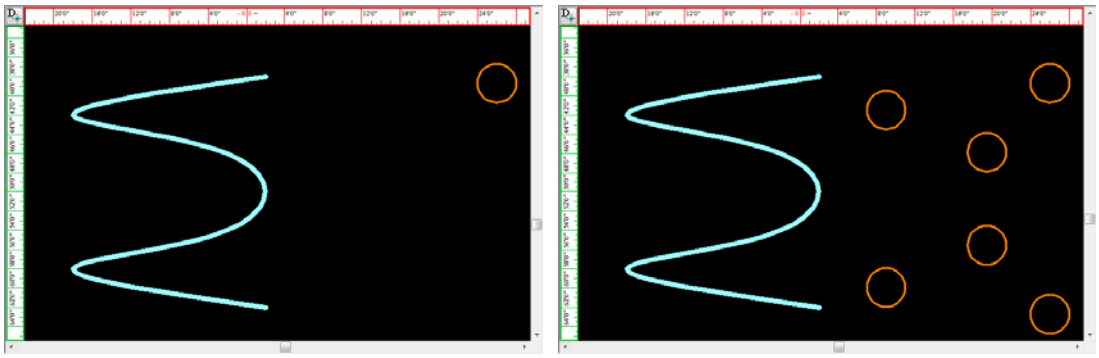


4 In the *Array Along Path* window, type the number of times you want the object copied.

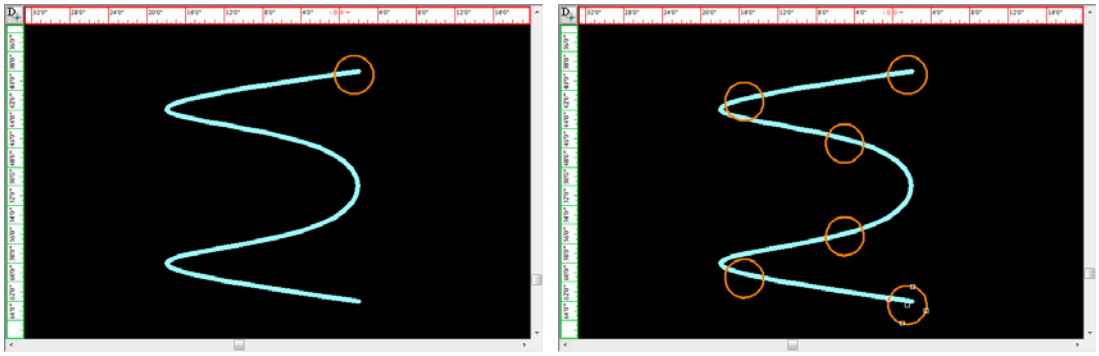
5 To have the object copies aligned to the path, mark the checkbox next to *Align To Path*.

6 Click *OK*.

Result: The object will be cloned in a patter based on the path.



An example of a non-snapped object using array along a path.



An example of a snapped object using array along a path.

To perform a polar array

1 Select the object that you want to array.

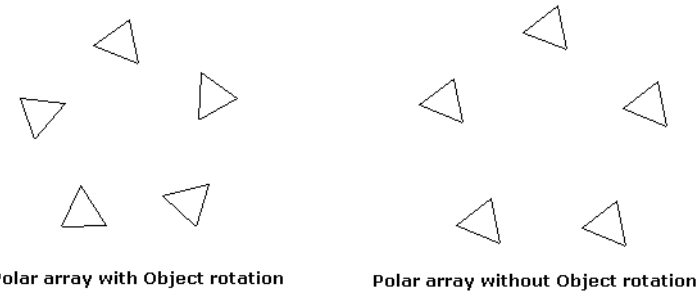
2 From the **Tools** menu, choose **Array**, and then choose **Array Polar**.

3 Click a point that will represent the center of the circle around which the objects will be arrayed.

4 In the dialog box that opens, type the total number of copies desired (this includes the original object).

5 Specify the total angle in degrees to fill with copied objects.

6 Select the *Rotate object* check box if you want the copied objects to rotate as they are pasted around the center of the circular array, as shown below:



Polar array with Object rotation

Polar array without Object rotation

7 Click *OK*.

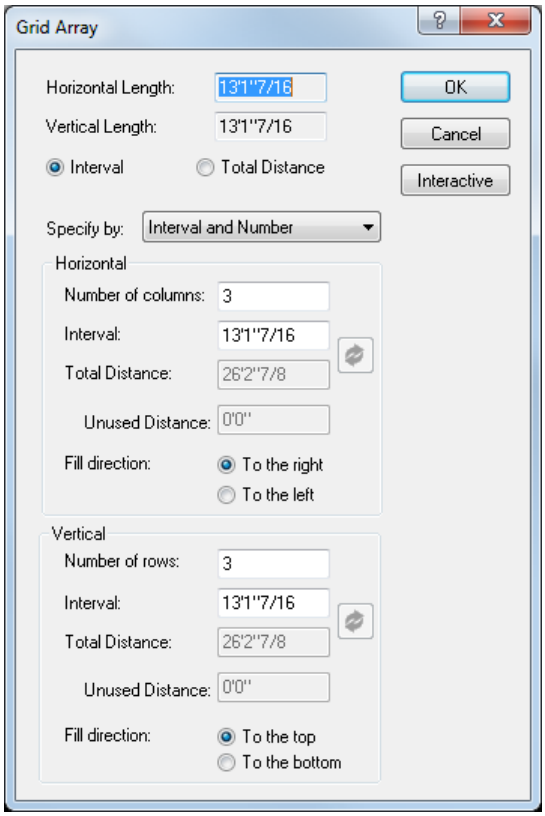
To perform a grid array

Perform this procedure to array the selected object in a grid pattern in the direction and at the distance that you specify.

1 Select the object that you want to array.

2 From the **Tools** menu, choose **Array**, and then choose **Array Grid**.

Result: The *Grid Array* window appears.



3 Select whether the horizontal and vertical lengths will be an *Interval*, or the *Total Distance* of the array.

Note: Click *Freehand* to manually draw the horizontal and vertical lengths of the gird array.

4 Specify which values will be available for editing.

Note: When performing an array, the value not chosen for editing is calculated automatically by WYSIWYG and grayed out.

Note: To undo changes made to the clone values, click the *Reset* button .

5 Enter the number of columns in the grid.

Note: This value includes the original object(s) as the first “column” of the grid. For example, if you type 3, the object is copied and pasted twice beside the original object in the direction and at the distance that you specify.

6 Enter the spacing between each horizontal object in the grid.

7 Enter the distance between each horizontal object in the grid.

8 Select the direction in which you want the object(s) to be copied horizontally in the grid, either to the right or left of the original object.

9 Type the number of rows in the grid.

Note: This value includes the original object(s) as the first “row” of the grid. For example, if you type 3, the object is copied and pasted twice above or below the original object at the distance that you specify.

10 Enter the distance between each vertical object in the grid.

11 Enter the distance between each vertical object in the grid.

12 Select the direction in which you want the object(s) to be copied in the grid vertically, either above or below the original object.

13 Click *OK*.

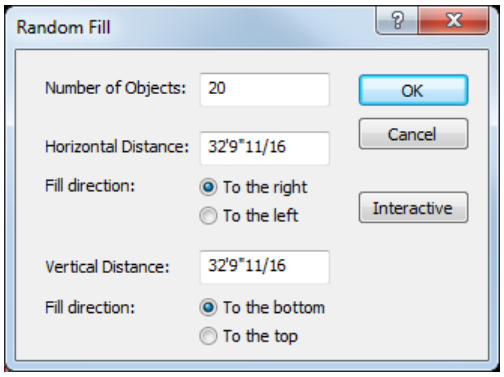
To perform a random fill

Random fill is used to take an object and randomly copy it in a defined space.

1 Select the object that you want to copy.

2 From the **Tools** menu, choose **Array**, and then choose **Random Fill**.

Note: The *Random fill* window appears.



3 Enter the number of copied objects to appear.

4 Enter the horizontal distance of the defined space.

Note: You can select *Interactive* to draw the horizontal and vertical lengths.

5 Select the direction in which you want the object(s) to be copied horizontally, either left or right the original object.

6 Enter the number of copied objects to appear.

7 Enter the horizontal distance of the defined space.

8 Select the direction in which you want the object(s) to be copied vertically, either above or below the original object.

9 Click *OK*.

Distribute Objects

A group of objects can be selected and then distributed in a straight line between two points.

To distribute objects between two points

1 Select the objects to be distributed.

From the **Tools** menu, **Distribute Objects** sub-menu, choose **Between Two Points.**

or

Click the *Between Two Points* tool on the *Modify* toolbar. 

2 Click on the drawing to set the first distribution point.

3 Click on the drawing to set the second distribution point.

Result: The objects will be distributed evenly in a straight line between the two chosen points.

Distance

The distance tool enables you to measure the distance and angle from one point to another.

To measure a distance

1 From the **Tools** menu, choose **Distance**.

Tip: You can also use the *Distance* tool on the *Tools* toolbar. 

2 Pick the starting point for the measurement.

3 Pick the end point of the measurement.

Result: The length and angle of the specified trajectory is displayed in the bottom left corner of the status bar.

Tip: When selecting the starting and end points for the measurement use the snap tools for precision.

Boolean Operations

The Boolean Operations series of tools enable you to choose two different sets of objects and then perform one of three actions on them:

- **Unite Objects** - This option enables you to join the two sets of objects together as one, merging their common area so that it is indistinguishable from the original objects.
- **Intersect Objects** - This option enables you to choose two sets of intersecting objects and then delete everything *outside* of their common area.
- **Subtract Objects** - This option enables you to select two sets of objects, and then subtract the overlapping area of the second set from the first set that you chose.

Selection sets

When using any of the Boolean Operations tools, you must select two sets of objects, **Set 1** and **Set 2**. Here are some guidelines for your selection sets:

- The only valid 2D object for Boolean operations is a surface.
- Valid 3D objects for Boolean operations include: Risers, Cylinders, Spheres, Extruded Surfaces, as well as any 3D objects resulting from a 3D x 3D Boolean operation. You cannot perform Boolean operations on library items or venues. To include a venue, you must first break it into surfaces.
- Each selection set can include one or more 2D objects. You cannot perform a Boolean operation on grouped objects. Instead, multiple objects can be selected at once by pressing CTRL on your keyboard.
- Each set can include only one 3D object.
- You cannot combine 2D and 3D objects in a single selection set.
- Each set can include either 2D or 3D objects. You can specify a selection set comprising one or more 2D objects and a selection set with one 3D object in the same Boolean operation.
- All operations that involve a 3D object require that Set 1 intersects with Set 2.
- For operations that involve only 2D objects, the objects in each selection set do not need to intersect physically; however, the *projection* of the objects in Set 2 must intersect with those in Set 1 along the plane in which you are currently working. For details, see “Projection and intersection” on page 242.
- In a Boolean operation involving one or more 2D objects in one selection set and a 3D object in the other selection set, the 3D object must be in the second selection set.

With these rules in mind, the possibilities for selection set combinations are:

- one 3d object x one 3d object
- one 2d object x one 2d object
- one 2d object x one 3d object
- one 2d object x multiple 2d objects
- multiple 2d objects x one 2d object
- multiple 2d objects x one 3d object
- multiple 2d objects x multiple 2d objects

Color and texture

Based on the type of object in your selection set, color and texture are treated differently after you perform a Boolean operation.

- 2D objects maintain their color and texture after a Boolean operation is performed on them; 3D objects maintain only their color (they lose any texture applied to them).
- In a Boolean operation involving only 2D objects or only 3D objects, the objects in the second selection set adopt the properties of the objects in the first selection set after the operation is complete.
- In a Boolean operation involving only 3D objects, if one or both of the objects has a texture applied, then both objects end up with no texture or color after the operation is complete.
- When you perform a Boolean operation on a 3D object, the object is converted into a set piece after the operation. Since you cannot apply a texture to set pieces, it is recommended that you convert it into a custom library object and then apply the texture. For details, see “To create a custom library item” on page 176. Alternately, you can convert the object into 2D surfaces before you perform the Boolean operation and then customize it.

Projection and intersection

When performing any of the Boolean Operations procedures on selection sets involving 2D and 3D objects, note the difference between projection and intersection.

- **Projection**: When you are working with *only* 2D objects, the objects in one selection set need only “project” upon the objects in the other set within the active view; that is, the objects in each set do not have to physically touch, but they must overlap in the view in which you are performing the action.
- **Intersection**: When you are working with only 3D objects or a mixture of 2D and 3D objects, the objects in each selection set must physically touch before you can successfully perform the Boolean Operations action, that is, they must intersect with each other.

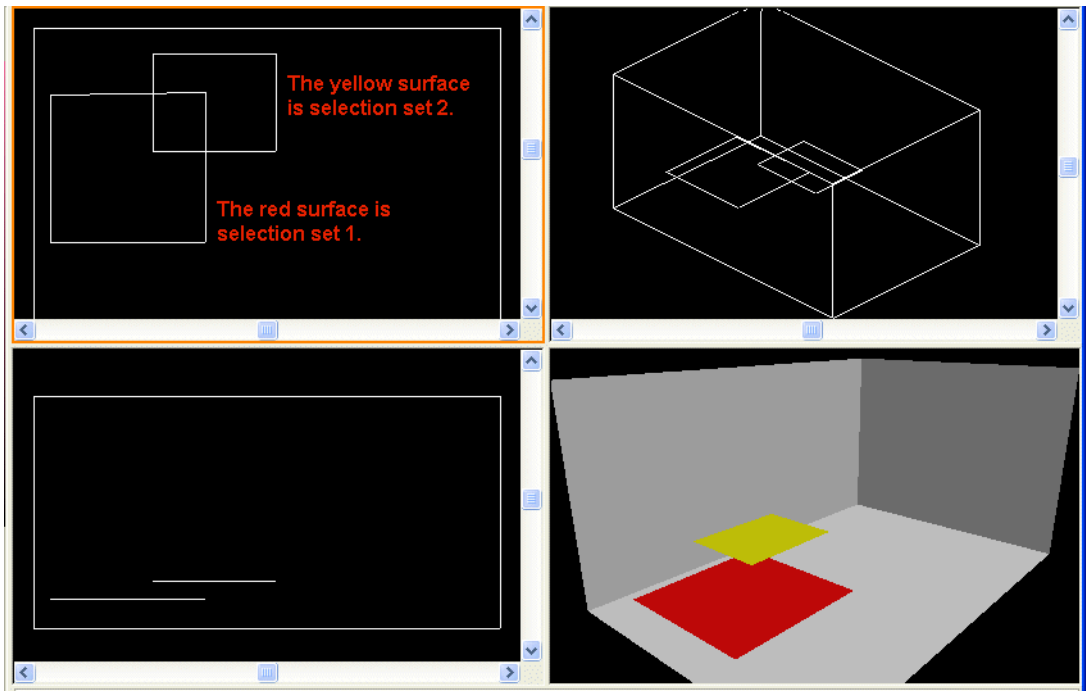
About the Boolean Operations feature and View Planes

There are three View Planes or axial planes in your drawing: (XY-plane, XZ-plane, and YZ-plane). Based on the view you are in (Plan, Front, Back, Left, or Right), some of these planes may not be visible. For example, when you are in Plan view, the XZ and YZ planes are invisible.

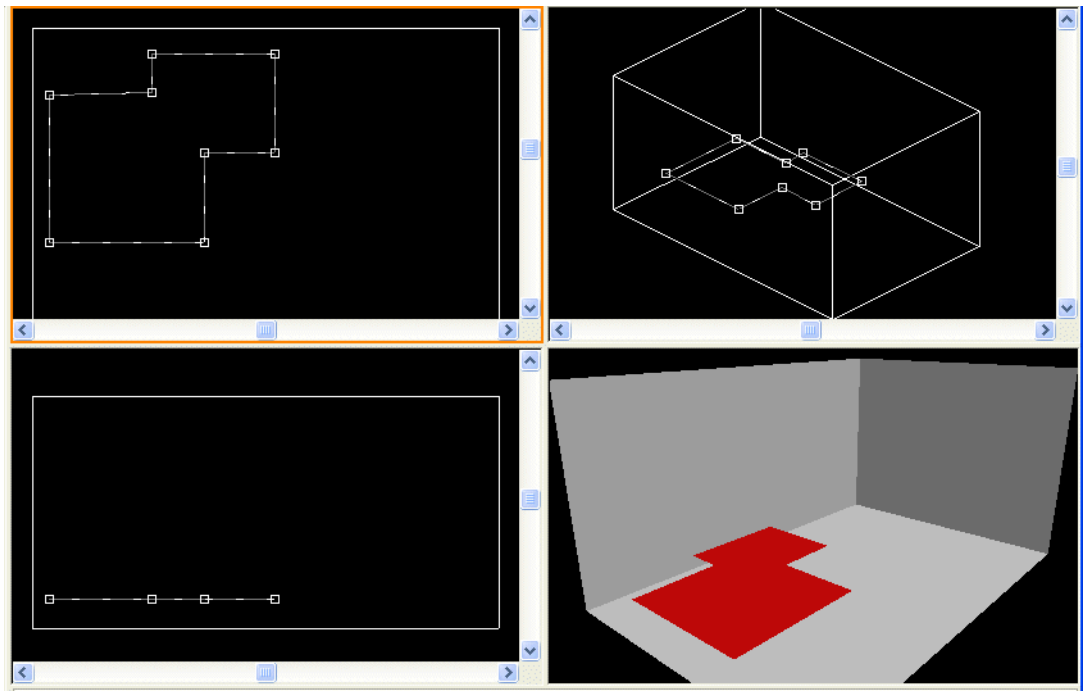
When you perform any of the Boolean Operations procedures with 2D surfaces in the second selection set, note how the objects appear on the active View Plane. Their appearance will give you an idea of the What-You-See-Is-What-You-Get result from the perspective of the active view. By noting the active View Planes, you can perform any of the Boolean operations on surfaces without requiring them to be co-planar.

Examples - 2D parallel objects

In the following graphic, selection set 1 includes a red 2D surface, and selection set 2 includes a yellow 2D surface. Both surfaces were drawn so that they overlap in Plan View, as shown in the upper-left quadrant:



When you perform a Unite procedure with these sets in the Plan view quadrant, the results are successful because you can see that the yellow surface projects onto the red surface in this view (they overlap).

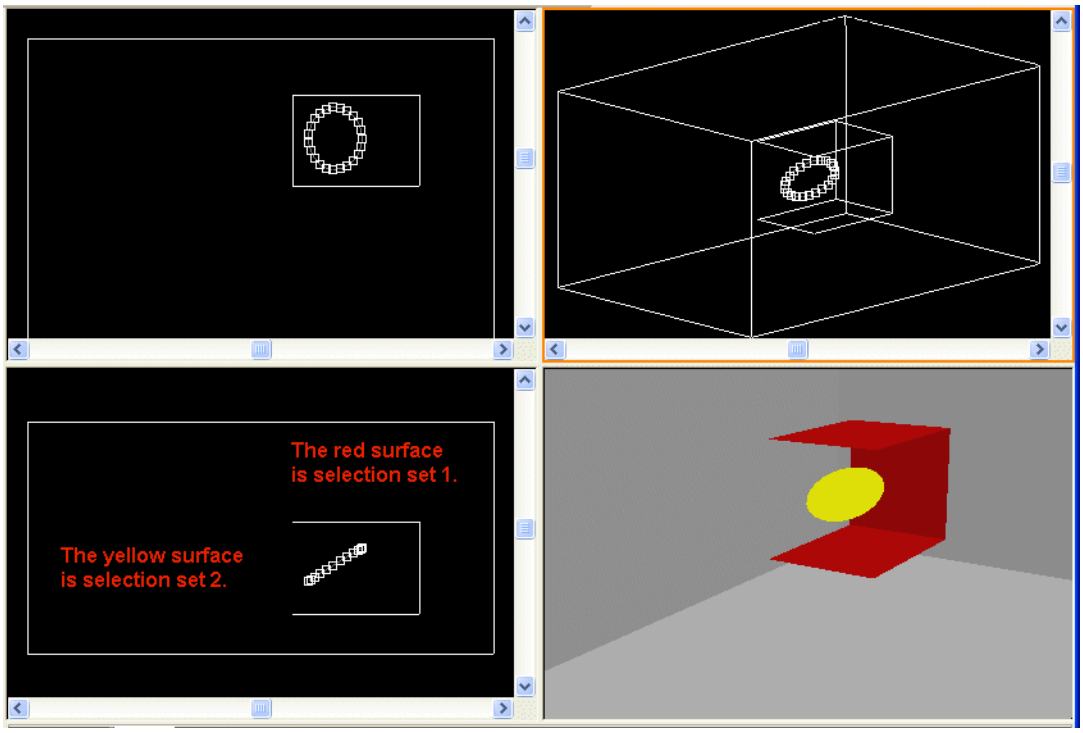


However, when you try to do the same action in the bottom left quadrant (Front View), the operation is unsuccessful because neither surface projects onto the other (they do not overlap in this view).

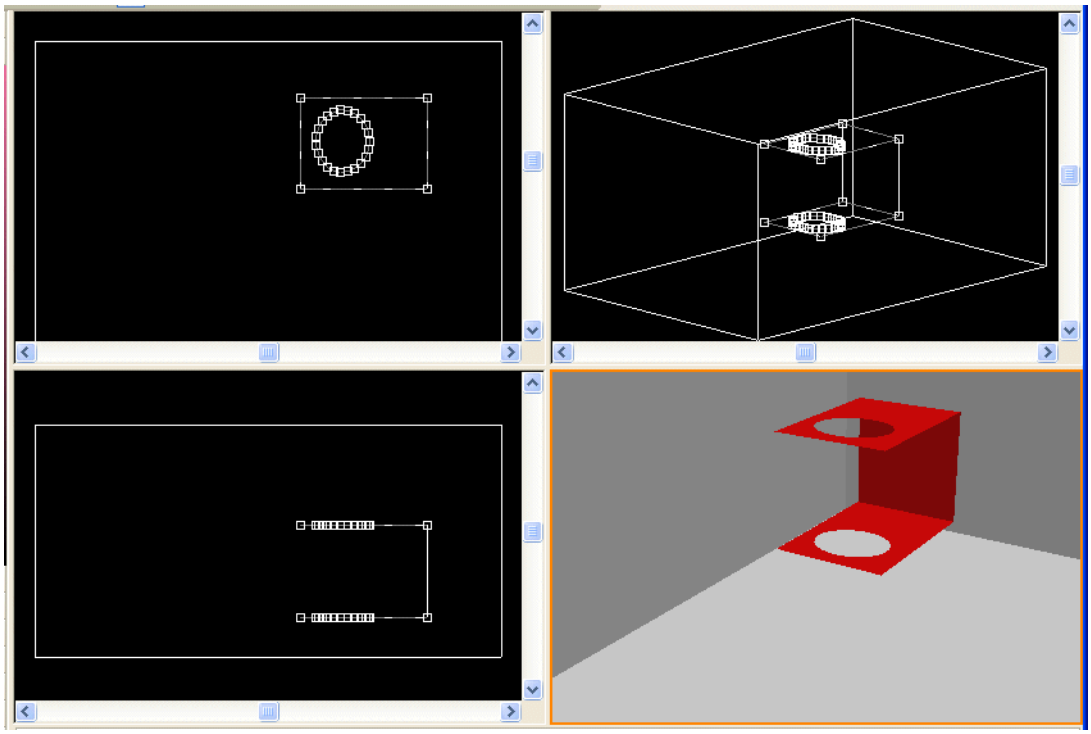
Tip:
To overcome this limitation, in the case noted above, you could extrude the surface in selection set 2 into a 3D solid, ensuring that the solid touches the surface in selection set 1. In some cases, it is much easier to perform a Boolean operation with 2D surfaces in the first selection set and a 3D solid in the second set because the operation is View Plane independent. For example, when you are cutting windows in walls, it is usually best to use 2D surfaces for both the windows and walls because you can project the image of the windows onto the walls (for details, see “To cut holes in walls with the Subtract tool” on page 250). However, if you have an odd-shaped building like a Pentagon, it would be impossible to align the walls with the View Plane. In this case, therefore, it would be easier to use cylinders for the shape of the windows and intersect them with the walls wherever you want a window to appear, before performing the Subtract procedure.

Examples - 2D non-parallel objects

If the surfaces in each of the selection sets are not parallel, the results of a Boolean operation can differ according to the active View Plane. In the following graphic, the red surfaces comprise selection set 1 and the yellow surface is selection set 2. Note that the yellow surface is rotated so that it is not parallel to any of the surfaces in set 1.

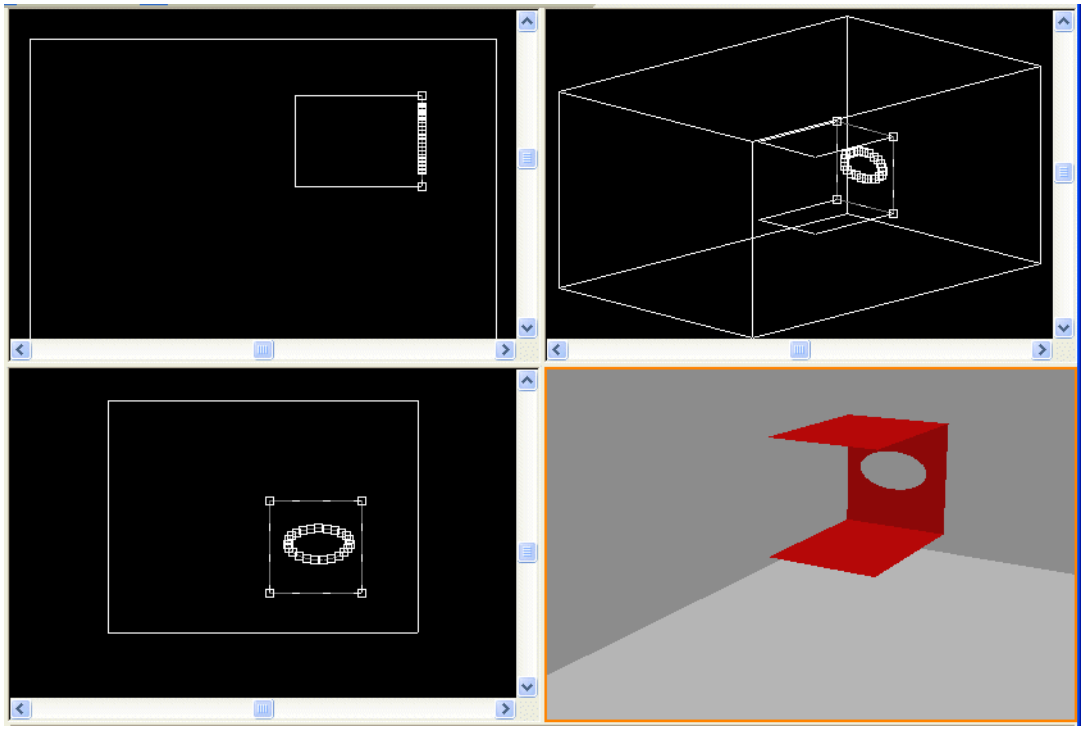


In this case, when you perform a Subtract operation in Plan view, the results are predictable: two holes the size of the yellow surface appear on each of the top and bottom red surfaces, as shown below. Note that the holes are not perfectly circular, but rather take on the exact shape of the angled circle as it appears in Plan view.



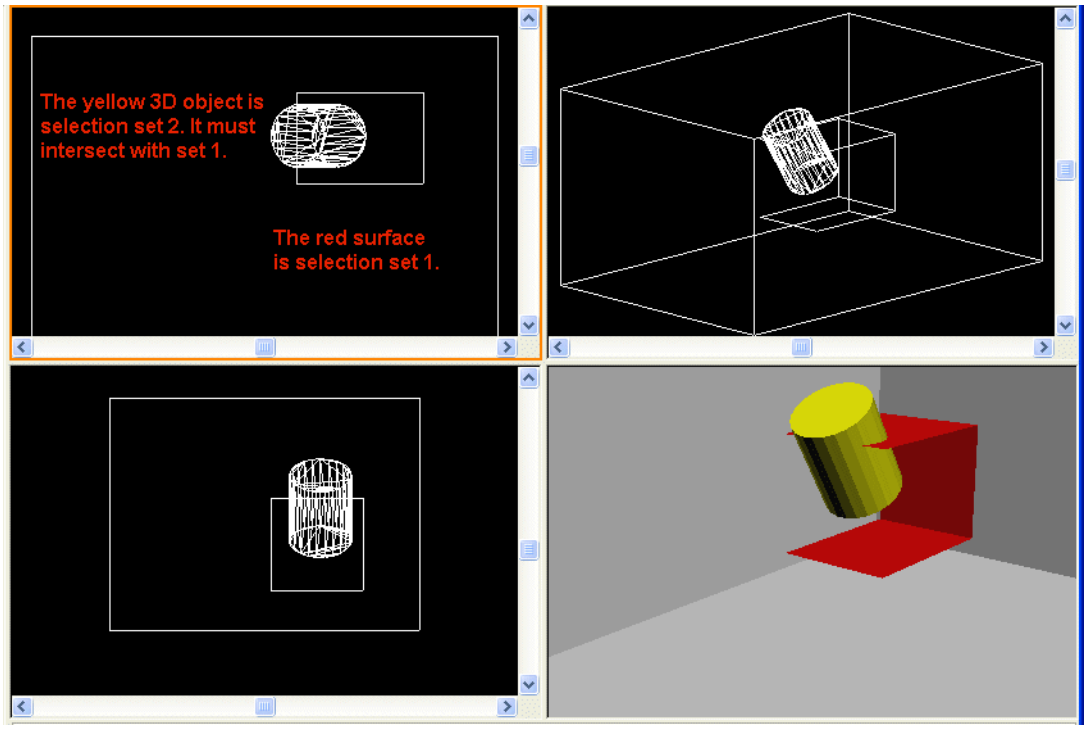
When you perform the same action in Front view, the operation fails because none of the surfaces in set 1 or set 2 is co-planar in this view.

However, when you switch to Right view and perform a Subtract, a hole appears only on the back surface (the one onto which the circle projects in this view), as shown below.

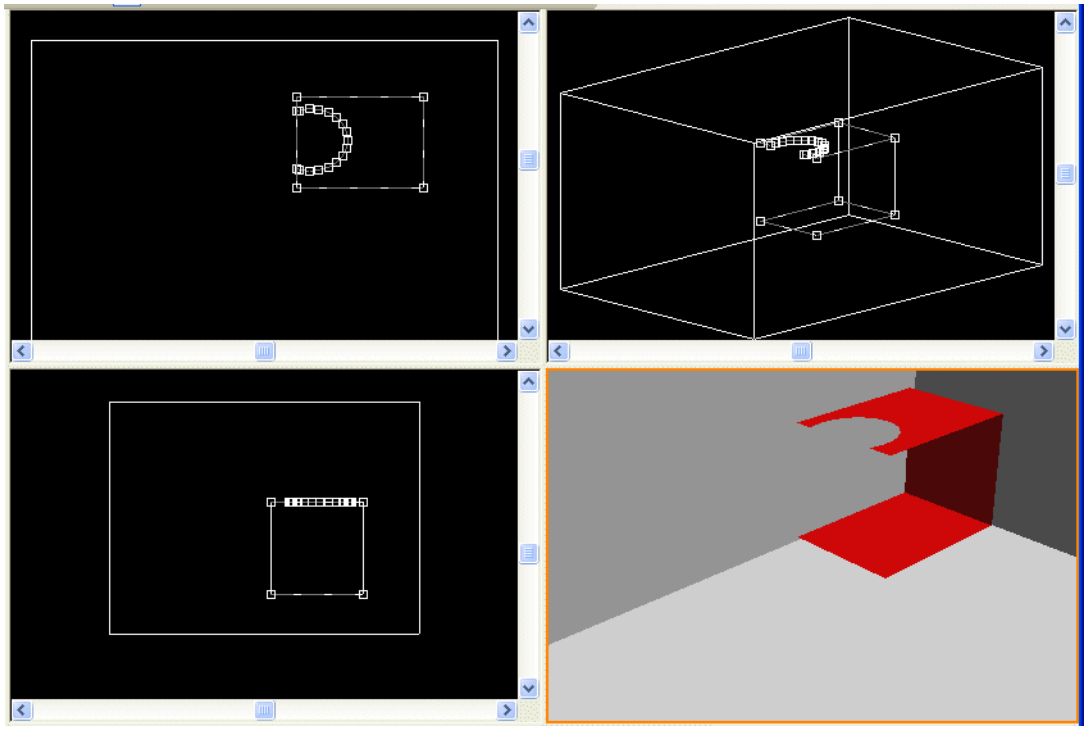


Example - 2D object and 3D solid

Note that if you convert the circular surface into a 3D solid, then you must ensure that it physically intersects with at least one of the 2D surfaces in set 1 before you can successfully perform a Boolean Operations procedure, as shown below; it is not enough for the object to project upon one of the surfaces. For details on projection and intersection, see “Projection and intersection” on page 242.



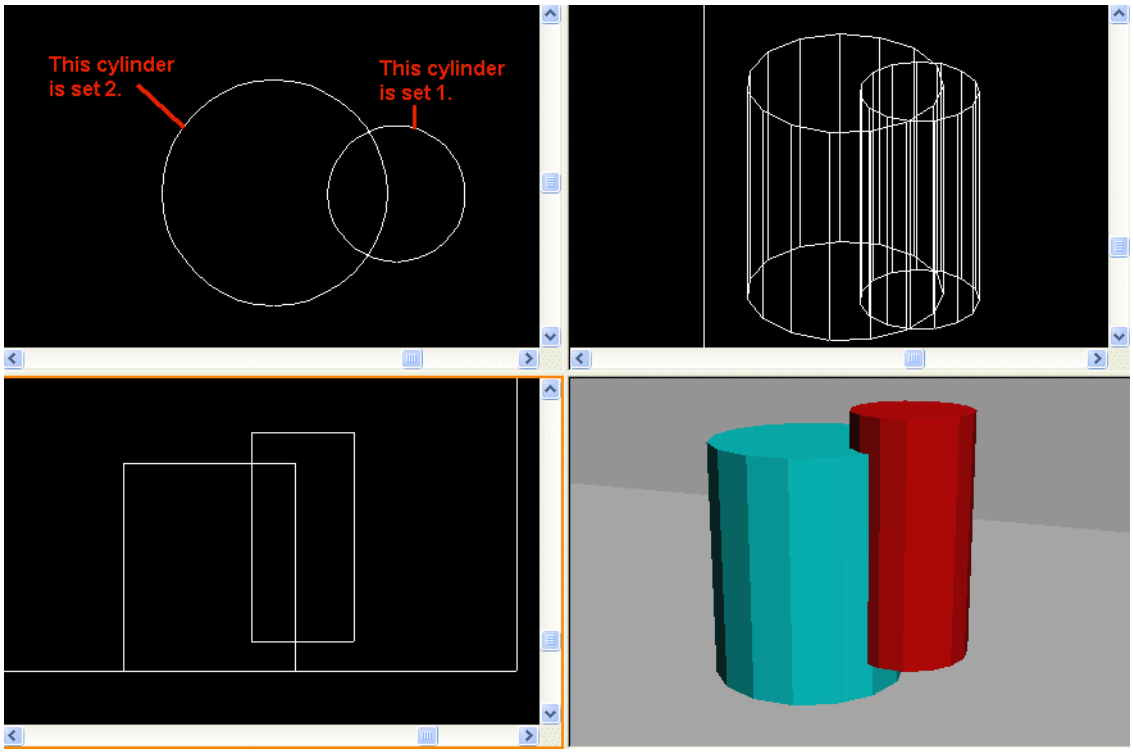
When you perform a Subtract action on these sets in Plan view, the results are as follows:



Examples of each Boolean Operations procedure

The following graphic shows the two object sets that will be used as examples in each of the procedures in this section.

Set 1 consists of the red cylinder and set 2 consists of the aqua cylinder:



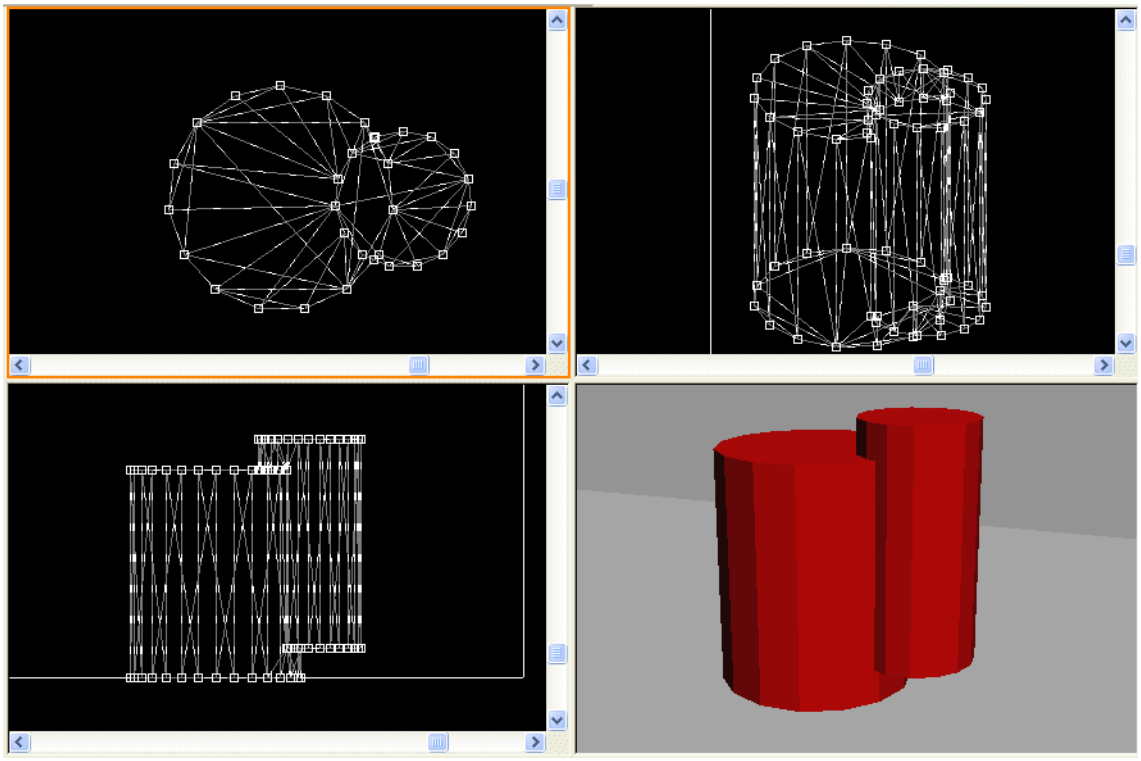
Uniting objects

This option enables you to join two sets of intersecting objects together as one, merging their common area. Note that when you unite objects, the second set of objects adopts the properties of the first set that you chose (i.e., its color, texture, etc.)

Note: If you are uniting a 2D object like a surface with a 3D object like a riser, the 2D object must be in the first selection set and the 3D object in the second. After you perform this operation, the 3D object adopts the properties of the 2D object, becoming a flat surface with the same color and texture as the 2D object.

The following graphic shows how the two sets of objects are united to become one object. Note how the second object set (the aqua cylinder) takes on the properties of set 1, becoming red.

Note: When you perform a Boolean operation on a 3D object, the object is converted into a set piece after the operation. Since you cannot apply a texture to set pieces, it is recommended that you convert it into a custom library object and then apply the texture. For details, see “To create a custom library item” on page 176. Alternately, you can convert the object into 2D surfaces before you perform the Boolean operation and then customize it.



To unite objects with the Boolean Operations feature

Note that you can only select multiple 2D objects in each selection set; you cannot select multiple 3D objects in a selection set.

1 In one of the 2D wireframe views of CAD mode, select the first object or objects that you want to join with the second object or objects.

2 Click **Tools** > **Boolean Operations** > **Unite Objects**.

3 Click to select the second object set that you want to join with first set.

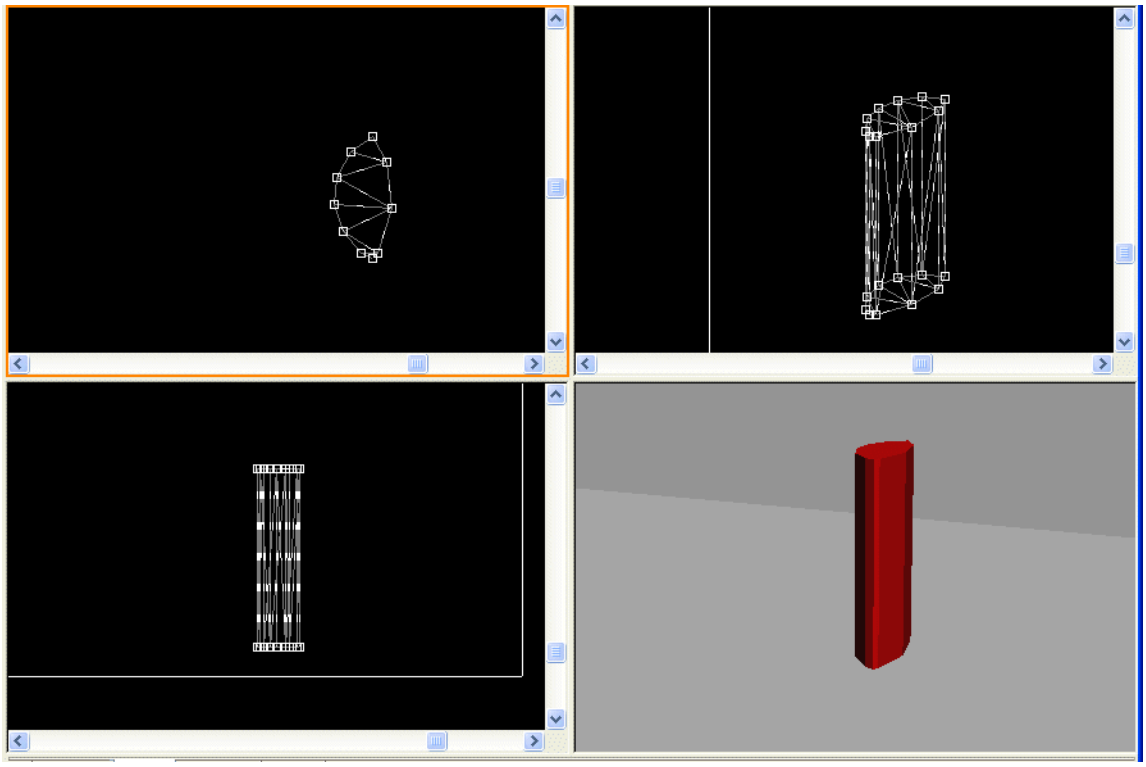
4 Right-click and choose **Finish Unite**.

Result: Based on the complexity of your selections, you may have to wait a few moments for the calculations to finish and the objects to be united.

Intersecting objects

This option enables you to choose two sets of overlapping or intersecting objects and then delete everything *outside* of their common area. Note that when you intersect objects, the intersection that remains behind maintains the properties of the objects in the first selection set.

The following graphic shows the result of intersecting the red and aqua cylinders. Note how the remaining portion is red, just like the cylinder in set 1.



To intersect objects with the Boolean Operations feature

Note that you can only select multiple 2D objects in each selection set; you cannot select multiple 3D objects in a selection set.

1. In one of the 2D wireframe views of CAD mode, select the first object or objects that you want to intersect with the second object or objects.
2. Click **Tools** > **Boolean Operations** > **Intersect Objects**.
3. Click to select the second object set that you want to intersect with first set.
4. Right-click and choose **Finish Intersect**.

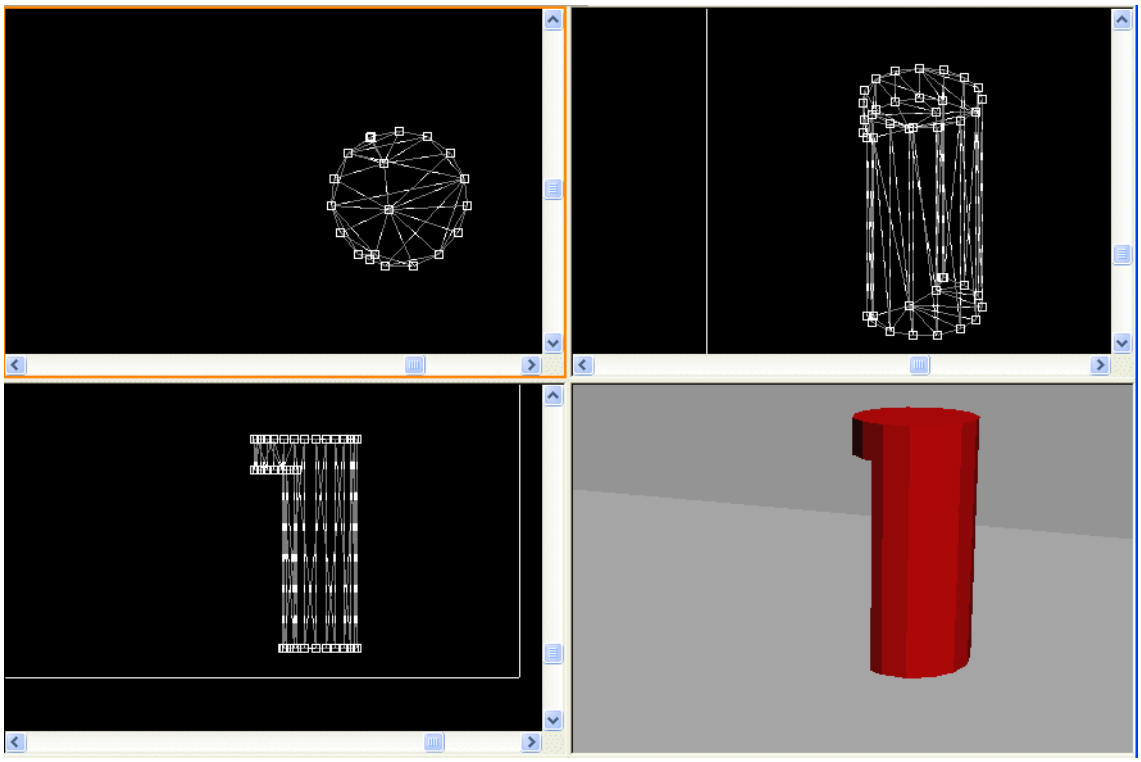
   Result: Based on the complexity of your selections, you may have to wait a few moments for the calculations to finish and the objects to be intersected.

Subtracting objects

This option enables you to select two sets of objects, and then subtract the overlapping area of the second set from the first set that you chose.

This feature is particularly useful for cutting a hole in a surface to create a window. Note, however, that if you are cutting a hole in a wall to create a window, it is easiest to work only with 2D objects in each selection set since they do not have to physically touch. Instead, the "window" shapes need only to project onto the walls in the active view. For details, see "Projection and intersection" on page 242. For details on cutting holes into walls, see "To cut holes in walls with the Subtract tool" on page 250.

The following graphic shows the result of subtracting the aqua cylinder from the red cylinder.



To subtract objects with the Boolean Operations feature

Note that you can only select multiple 2D objects in each selection set; you cannot select multiple 3D objects in a selection set.

1 In one of the 2D wireframe views of CAD mode, select the first object or objects that you want to keep in your drawing (the second set of objects will be subtracted from this set).

2 Click **Tools** > **Boolean Operations** > **Subtract Objects**.

3 Click to select the second object set that you want to subtract from the first set.

4 Right-click and choose **Finish Subtract**.

Result: Based on the complexity of your selections, you may have to wait a few moments for the calculations to finish and the objects to be subtracted.

To cut holes in walls with the Subtract tool

In this procedure, you use the Boolean Operations > Subtract tool to cut a line of round windows into two opposing walls of your venue.

1 In the CAD Mode > Drawing Quad tab, draw a venue (for example, a black box).

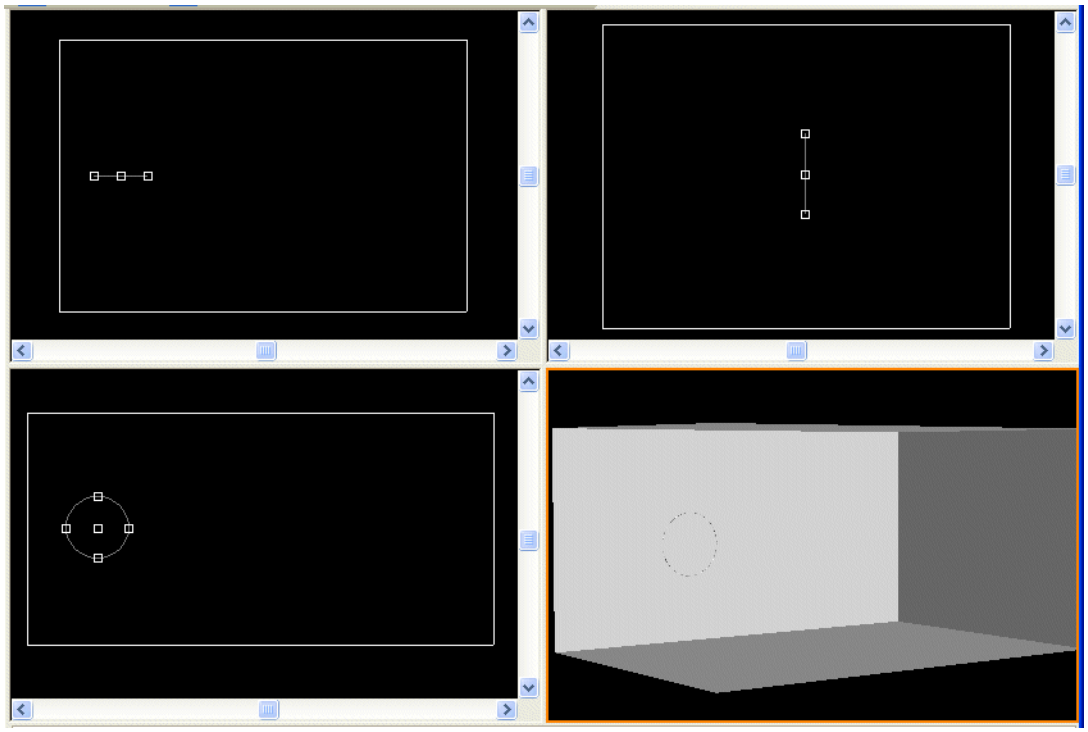
2 Click the venue to select it, and then right-click and choose **Tools** > **Break** > **Solids into Surfaces**.

Note: Since the window shapes are a series of 2D surfaces, it is best to convert the 3D venue into a 2D surface as well. This way, the window shapes only have to project onto the "walls" in the same plane. If the venue is a 3D object, then the windows would have to physically touch them (intersect) before you perform the Subtract operation. For details, see "Projection and intersection" on page 242.

3 Click to highlight the Front view quadrant.

4 Click the Circle tool, and then click *OK* to accept the default size of 4'.

5 In the Front view quadrant, click to place the circle half-way up the wall, at one end of the room, as shown below:



6 With the circle still selected, right-click and choose **Tools** > **Convert** > **Circles into Surfaces**.

7 Type **20** and click *OK*. (The higher the number of points, the smoother the outline of the circle.)

8 With the circle still selected, click **Tools** > **Array** > **Linear**.

9 In the Front view quadrant, click the extreme right-hand wall to pick the destination point.

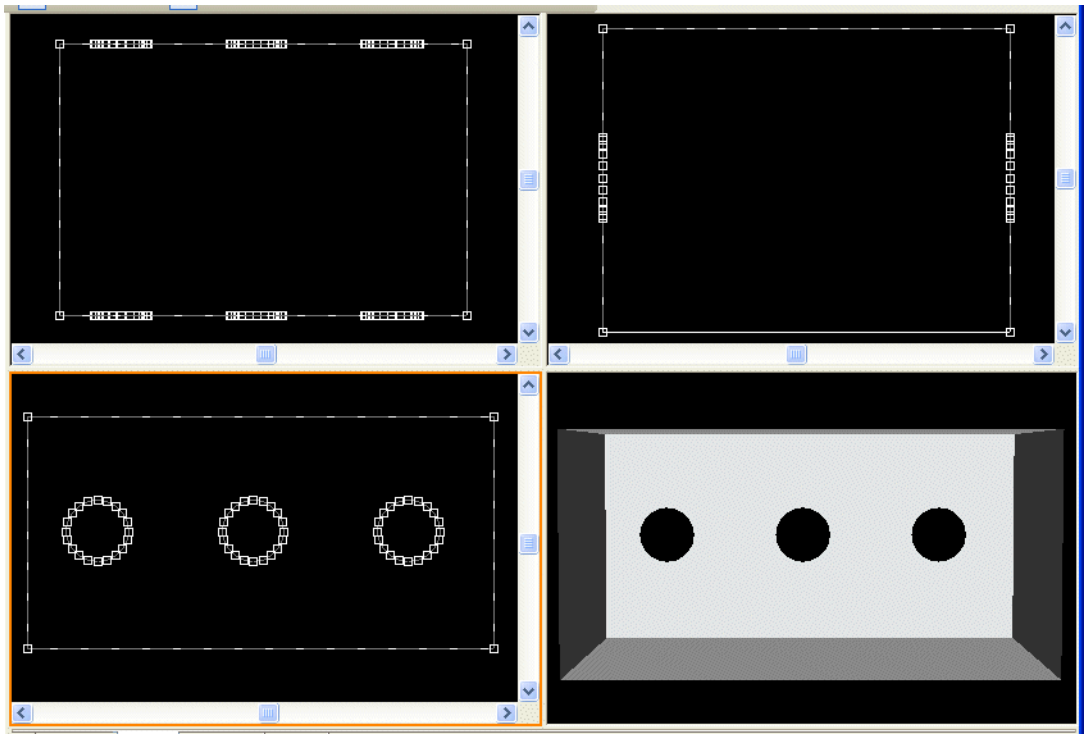
10 Type **3** for the Number in row and **20** for the Interval, and then click *OK*.

Result: The circular surfaces are arrayed along the length of the room.

11 In the Front view quadrant, click the Black Box outline to select it, and then click **Tools** > **Boolean Operations** > **Subtract Objects**.

12 Click to select all three circular surfaces at once, and then right-click and select **Finish Subtract**.

Result: The circular windows are “cut” into the walls of the venue, as shown in the following graphic:



Consolidated Mesh

This feature helps increase performance in Shaded views by enabling you to consolidate complex objects comprising multiple polygons/”sub-objects” into objects that behave as a single entity. Consolidating meshes is essentially the same as creating new (custom) Library Items. However, while both methods make WYSIWYG “see” the object as a single entity, when you use the Consolidate Mesh command, you don’t actually create a new Library Item, so your file size remains almost the same.

To determine which objects in your scene to consolidate, it is recommended that you enable the new *Object Bounding Box* feature on the View Options tab. Once you do so, objects that appear with a high number of bounding boxes are all candidates for consolidation.

Note: If you consolidate an object to which a texture has been applied, the texture may not behave correctly because the object’s underlying geometry will be different. To maintain proper texturing AND have the file perform well for pre-visualization, it is recommended that you create a copy of the final file. Optimize the copy, and then use the optimized file for pre-programming purposes and the original file for screenshots and/or renderings.

To use the Consolidated Mesh feature

Note: Boolean Operations cannot be performed on Consolidated Mesh objects.

1 In a wireframe view, select the object that you want to consolidate.

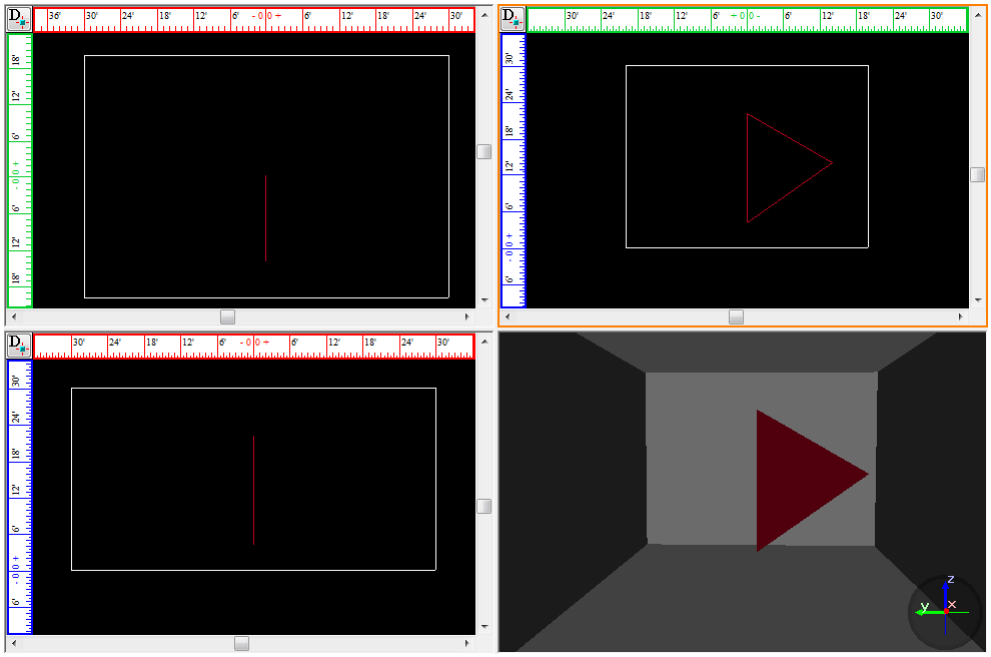
2 Click **Tools > Consolidate Mesh**.

Result: A message box appears, confirming that the operation is complete, and displaying the number of polygons that have been removed from the file.

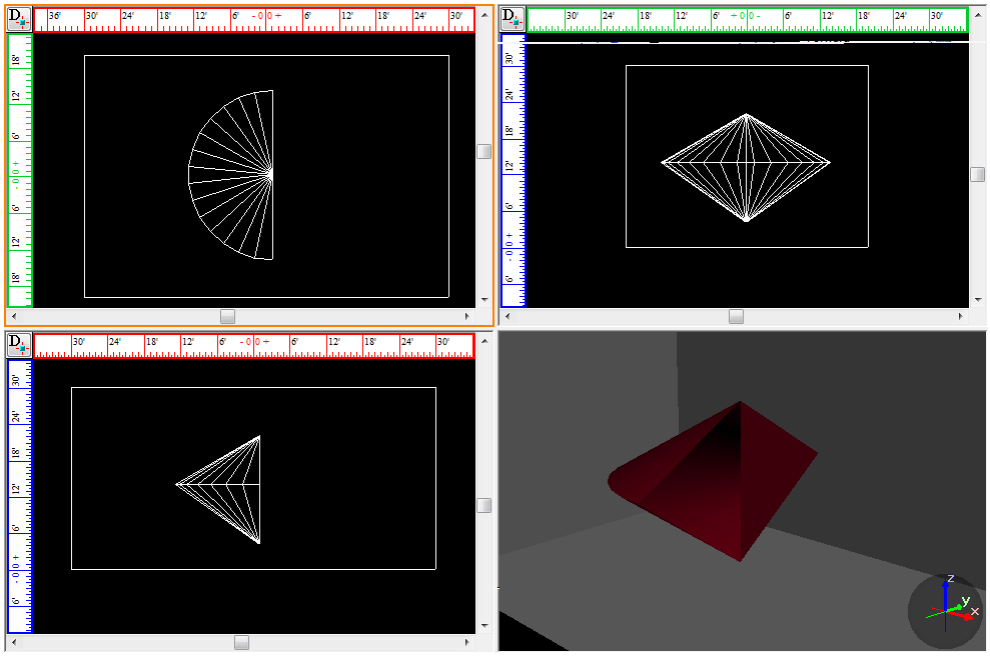
3 Click *OK*. The item is now consolidated into an abject that behaves as a single entity.

Revolve

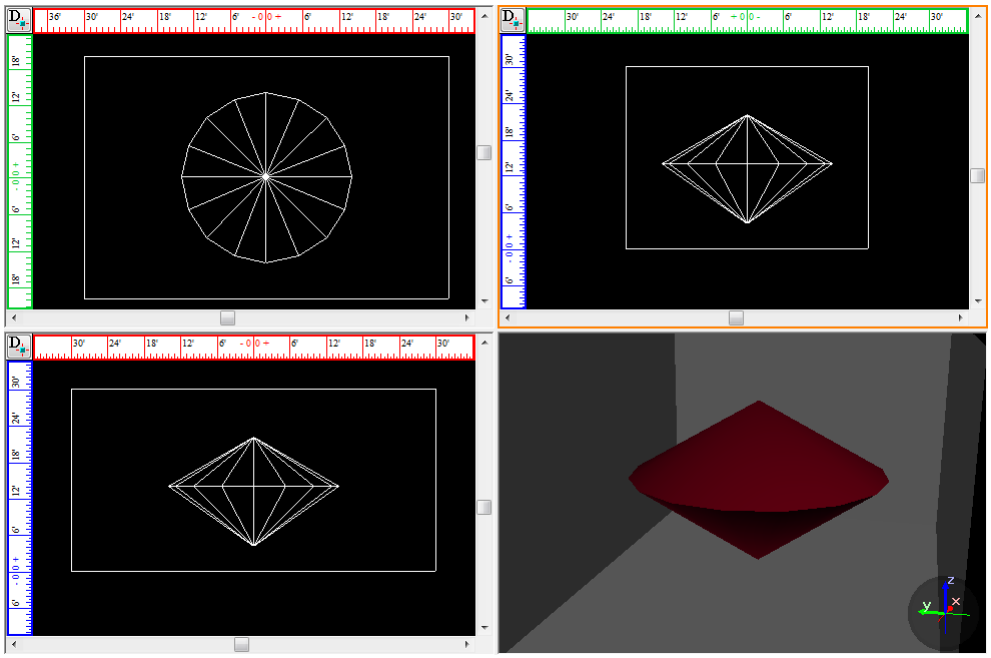
This feature enables the creation of three-dimensional object by revolving a surface along an axis, copying the surface X number of times along the rotation, and filling in the spaces in between to make it solid.



An example simple surface before using the revolve feature.



The example surface revolved 180° from the axis.



The example surface revolved 360° from the axis.

To use the Revolve feature

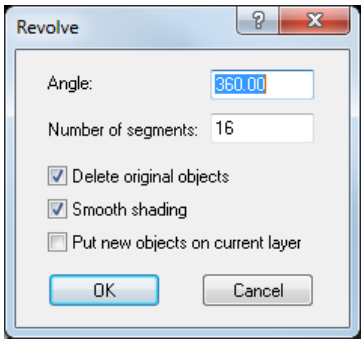
1 In a wireframe view, select the surface that you want to revolve.

2 Click **Tools > Revolve**.

3 Click to place the first axis point the surface will revolve around.

4 Click to place the second axis point the surface will revolve around.

Result: The *Revolve* window will appear.



5 Enter in how many degrees the surface will rotate on the axis in the **Angle** field.

6 Enter the number of times the surface is to be copied in the **Number of segments** field.

7 To remove the original surface after revolving, mark the checkbox by **Delete original objects**.

8 To smooth the revolved surface, mark the checkbox by **Smooth shading.**

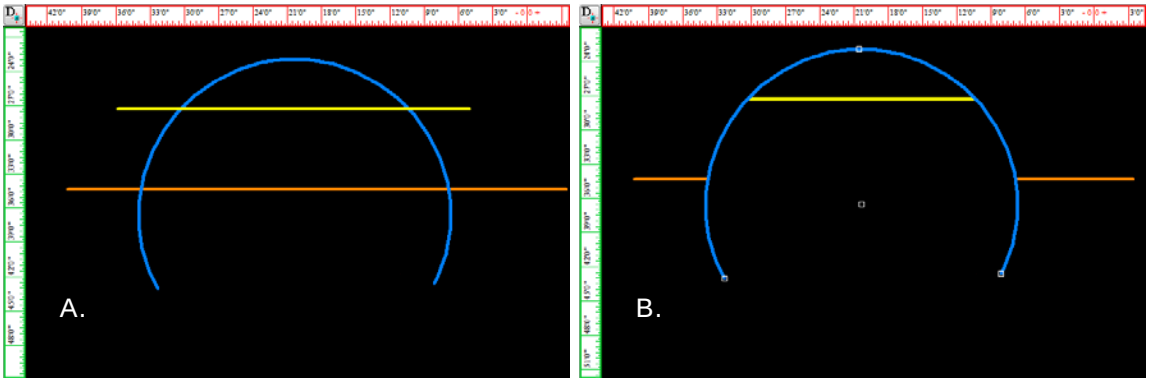
9 To include the revolved surface on the current layer, mark the checkbox by **Put new objects on current layer.**

10 Click *OK*.

Result: The surface will be revolved and create a three-dimensional object.

Trimming lines

The Trim tool can be used to remove sections of drawn lines that intersect a referenced object (arc, circle or line) drawn on the same plane. Trimming will remove all of the selected line segment, stopping at where it intersects with the referenced object.



An arc used as a reference object (image A.) with intersecting lines trimmed (image B.).

To trim a line

1. In the drawing, select the reference object.
2. From the **Tools** menu, choose **Trim**.

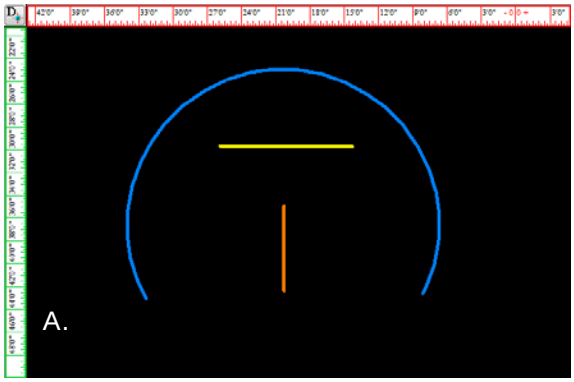
   Tip: The keyboard shortcut to access the Trim tool is CTRL+T.
3. Click on a segment of line you want to remove that intersects the referenced object.

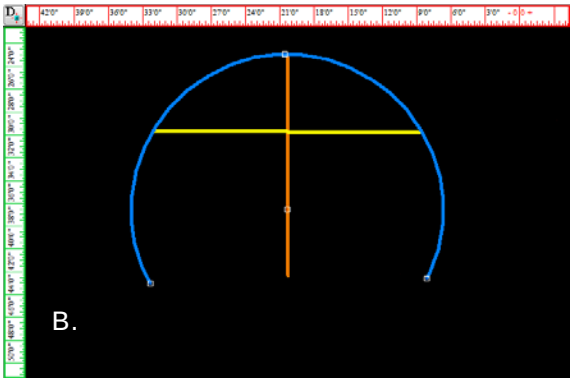
   Result: The selected line segment will be removed, from where the line intersects the reference object to where the line ends or intersects the object again.
4. Continue trimming lines as needed.
5. To end trimming line, right-click and choose **Finish Trim** or **Abort Trim**. Alternately, the ESC key can be pressed.

Extending lines

The Extend tool can be used to lengthen straight lines until they touch a referenced object (arc, circle or line) drawn on the same plane.

Note: Lines can only be extended if they are able to intersect the reference object when extended. If they will not intersect, the line will not extend.





An arc is used as a reference object (image A.). Other lines are extended until they intersect with the arc (image B.).

To extend a line

1 In the drawing, select the reference object (line, arc or circle).

2 From the **Draw** menu, choose **Extend**.

Tip: The keyboard shortcut to access the extend tool is CTRL+SHIFT+T.

3 Click on a line that you want to extend to the reference object.

Result: If the selected line can intersect the reference object, it will extend until it has done so. If the line can not intersect the object, it does not extend. The selected line will intersect all other objects in its path to reach the reference object. If the line can intersect the reference object on both ends, both line ends will extend until they intersect the reference object.

4 Continue extending lines as needed.

5 To end extending lines, right-click and choose **Finish Extend** or **Abort Trim**. Alternately, the ESC key can be pressed.

Hang structures

Introduction

Hang structures are a very important part of creating a lighting plot in WYSIWYG. All fixtures must be placed on a hang structure, which can be either a pipe or a truss; further, truss hang structures are ACTUAL pieces of truss, ladders, booms, floor mounts or stands. Hang structures are not only a place to hang fixtures, but also a means to organize your paperwork. Some general facts about hang structures:

- Pipes are drawn, trusses are inserted from the truss library. There are many different types of truss pieces in the truss library, including box, triangle, and curved.
- AutoUnit will assign and adjust unit numbers for fixtures on a pipe as you add, move, or delete them. AutoUnit does not work with truss.
- Hang structures can have the same name, even if they are not connected. For example, you can use this feature for set mounts, where you have many small pipes attached to the same piece of scenery.

Position Manager

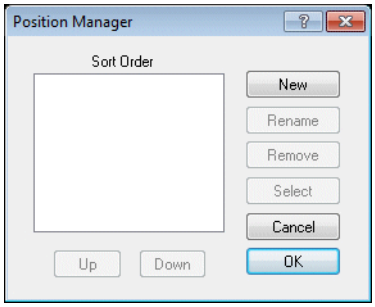
The Position Manager organizes all the available position names in your show document. It also sets the sort order of your positions. You can use the Position Manager to change the sort order of the position names, which is important for printed paperwork like instrument schedules and other reports that use the Position field. The order of position names is also used for general sorting in the Data view and the order in which positions appear in the Flight Case.

The Position Manager is accessible from the *Properties* window of a hang structure and from the **Positions** command in the **Edit** menu.

To view and edit entries in the Position Manager

1 From the **Edit** menu, choose **Positions**.

Note: The ellipsis button [...] on the Hang Structure page of a pipe's or truss' properties will also open the Position Manager.



2 Click *New*.

3 Type the name of the new position, and then click *OK*.

4 To modify the name of a position, select the position, and then click *Rename*.

5 Type the new name, and then click *OK*.

6 To remove a position name, select the name, and then click *Remove*. Removing the position name does not delete the hang structure. Rather the hang structure removes its association with the position name.

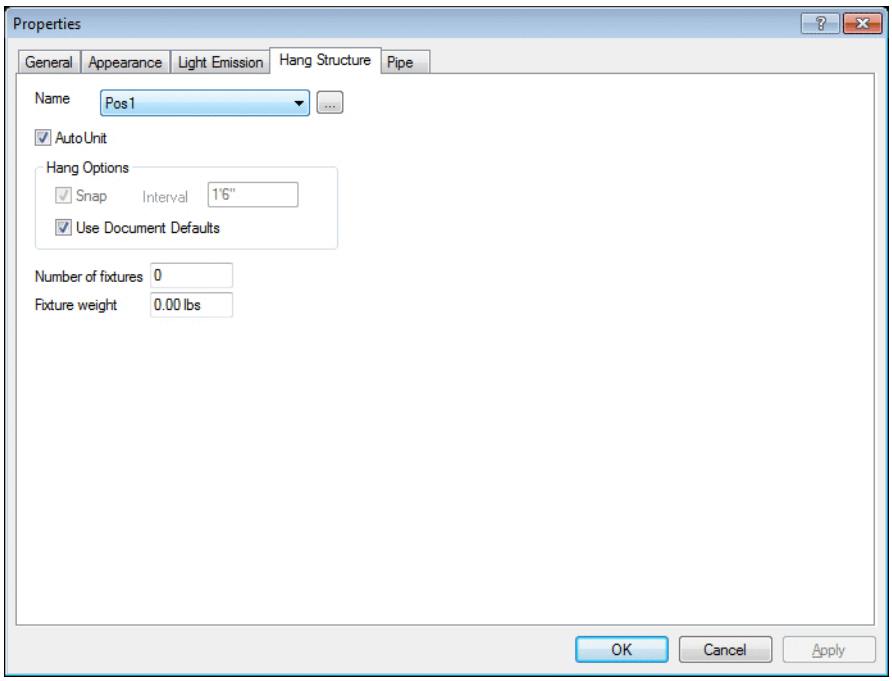
7 To reorder the position names in the list, select the name of the position you want to move. Click *Up* or *Down* to move the name in the list.

8 If you are in a plot view and want to determine what a particular position references, click *Select*. If the plot view is open (and the correct layers are marked editable), the hang structures that use that position name are selected. To select more than one position, press CTRL, and then click the position.

9 Click *OK* to close the Position Manager.

Hang structure properties

Options on the *Hang Structure* tab affect the properties of the selected hang structure(s).



- *Name:* Select a new name for the hang structure from the *Name* drop-down list. Click the ellipsis (...) button to open the Position Manager and create new names.
- *AutoUnit:* Select this check box to automatically assign a number to each fixture that you hang. Fixtures are automatically renumbered if you add a new fixture into the middle of a row of fixtures. This option is available for pipes only and does not apply to truss.
- *Snap:* Select this check box to hang fixtures on the pipe at a certain interval, as specified in the *Interval* box.
- *Interval:* Type the distance between fixtures in this box. This option is used for pipe snaps.
- *Use Document Defaults*: Select this check box to use the interval for pipe snap as defined on the *Object settings* tab in *Show Options*. For more information on this tab, refer to “Object Settings tab” on page 398.
- *Number of fixtures:* The number of fixtures on the hang structure.
- *Fixture weight:* The total weight of fixtures on the hang structure. The weight measurement accounts for fixture weight only and does not consider cable or other safety ratings.

Drawing pipes

Pipes are single member hang structures. Pipes can be set to be drawn from the center or from the start.

The first pipe drawing mode, Draw from Start, requires pipes to be drawn by specifying the start point and the end point (from one extremity of the pipe to the other).

The second pipe drawing mode, Draw from Center, requires that the pipe be drawn by specifying the center point and one end point or extremity of the pipe. Although it is only necessary to enter all points when drawing in interactive mode, the pipe drawing mode will affect the insertion point of the pipe when drawing using the pipe dialog box. The drawing mode will also affect how the fixtures are hung when using pipe snap, how the fixture offset distance is measured, and how pipe tape prints are labelled.

To set the pipe drawing mode

1 From the **Options** menu, choose **User Options**.

2 Click the *User* tab.

3 In the *Pipe Drawing Mode* section, select *Draw from Center* or *Draw from Start*.

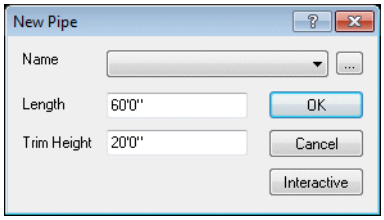
Tips:

- You can change the default setting for a pipe in mid-command by right-clicking and choosing the drawing mode for that pipe just before inserting it into the drawing.
- You can change the pipe mode for a specific pipe at any time in the pipe's properties. For more information, refer to "Pipe properties" on page 262.

To draw a pipe

1 From the **Draw** menu, choose **Pipe**.

Result: The *New Pipe* window opens.



2 In the *New Pipe* dialog box, find the name of the pipe. If you need to create the name of the pipe, click the ellipsis button [...] to open the Position Manager. Click *New*, and then type the name of the pipe.

3 In the *Length* box, type the length of the pipe, if different from the default. The default value is 60'0".

4 In the *Trim Height* box, type the desired trim height for the pipe. The default value is 20'0".

Note: Pipes ignore the missing coordinate setting when drawn using this method.

5 Click *OK*.

Tip:

You may also use the *Pipe* tool on the *Draw* toolbar. 

To draw a pipe from center in interactive mode

1 From the **Draw** menu, choose **Pipe**.

Result: The *New Pipe* window opens.

2 In the *New Pipe* dialog box, click *Interactive*

3 Set the height of the pipe that you are about to draw by pressing the TAB key, and then entering a value for the missing coordinate, if applicable.

4 Right click and select **Pipe - Draw from center**.

5 Click to place the center point of the pipe.

6 Drag it to the proper length, and then click to place the end point of the pipe. Notice that the pipe extends in both directions - mirrored at center.

7 Right-click on the pipe that you have just drawn, and then click **Properties**.

8 Click the *Hang Structure* tab.

9 From the Name drop-down list, select the name for this new pipe. If the name is not already in the list, use the *Position Manager* [...] to enter the new name.

10 Repeat to draw more pipes.

To draw a pipe from start in interactive mode

1 From the **Draw** menu, choose **Pipe**.

Result: The *New Pipe* window opens.

2 In the *New Pipe* dialog box, click *Interactive*

3 Set the height of the pipe that you are about to draw by pressing the TAB key, and then entering a value for the missing coordinate, if applicable.

4 Right click and select **Pipe - Draw from start**.

5 Click to place the starting point of the pipe.

6 Drag it to the proper length, and then click to place the end point of the pipe. Notice that the pipe extends in one direction.

7 Right-click on the pipe that you have just drawn, and then click **Properties**.

8 Click the *Hang Structure* tab.

9 From the Name drop-down list, select the name for this new pipe. If the name is not already in the list, use the *Position Manager* [...] to enter the new name.

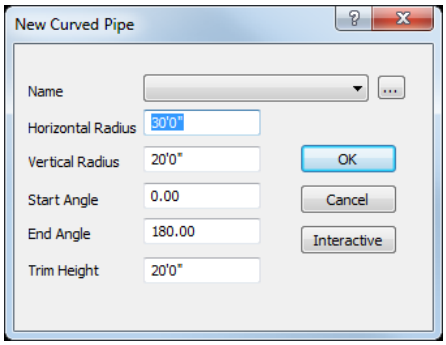
10 Repeat to draw more pipes.

Drawing curved pipes

In addition to drawing pipes linear in shape, pipes may be drawn curved in arcs.

To draw a curved pipe

1 From the **Draw** menu, choose **Curved Pipe**.



Result: The *New Curved Pipe* window will appear.

2 In the *New Curved Pipe* dialog field, find the name of the pipe. If you need to create the name of the pipe, click the ellipsis button [...] to open the Position Manager. Click *New*, and then type the name of the pipe.

3 In the *Horizontal Radius* field, type how long the pipe radius will be horizontally, if different from the default. The default value is 30'0".

4 In the *Vertical Radius* field, type how long the pipe radius will be vertically, if different from the default. The default value is 30'0".

5 In the *Start Angle* field, type what angle the pipe orientation will start at, if different from the default. The default value is 0°.

6 In the *End Angle* field, type what angle the pipe orientation will end at, if different from the default. The default value is 180°.

7 In the *Trim Height* box, type the desired trim height for the pipe. The default value is 20'0".

Note: Pipes ignore the missing coordinate setting when drawn using this method.

8 Click *OK*.

To draw a curved pipe from center in interactive mode

1 From the **Draw** menu, choose **Curved Pipe**.

Result: The *New Curved Pipe* window will appear.

2 In the *New Pipe* dialog box, click *Interactive*

3 Set the height of the pipe that you are about to draw by pressing the TAB key, and then entering a value for the missing coordinate, if applicable.

4 Right click and select **Pipe - Draw from center**.

5 Click to place the first point of the horizontal axis.

6 Click to place the second point of the horizontal axis.

7 Click to set the vertical radius.

8 Click to place the center point of the pipe.

9 Drag it to the proper length, and then click to place the mid point of the pipe.

10 Click to place the start point of the pipe. Notice that the pipe extends in both directions - mirrored at center.

11 Right-click on the pipe that you have just drawn, and then click **Properties**.

12 Click the *Hang Structure* tab.

13 From the Name drop-down list, select the name for this new pipe. If the name is not already in the list, use the *Position Manager* [...] to enter the new name.

14 Repeat to draw more pipes.

To draw a curved pipe from start in interactive mode

1 From the **Draw** menu, choose **Curved Pipe**.

Result: The *New Curved Pipe* window will appear.

2 In the *New Pipe* dialog box, click *Interactive*

3 Set the height of the pipe that you are about to draw by pressing the TAB key, and then entering a value for the missing coordinate, if applicable.

4 Right click and select **Pipe - Draw from start**.

5 Click to place the first point of the horizontal axis.

6 Click to place the second point of the horizontal axis.

7 Click to set the vertical radius.

8 Click to place the center point of the pipe.

9 Drag it to the proper length, and then click to place the mid point of the pipe.

10 Click to place the start point of the pipe. Notice that the pipe extends in both directions - mirrored at center.

11 Right-click on the pipe that you have just drawn, and then click **Properties**.

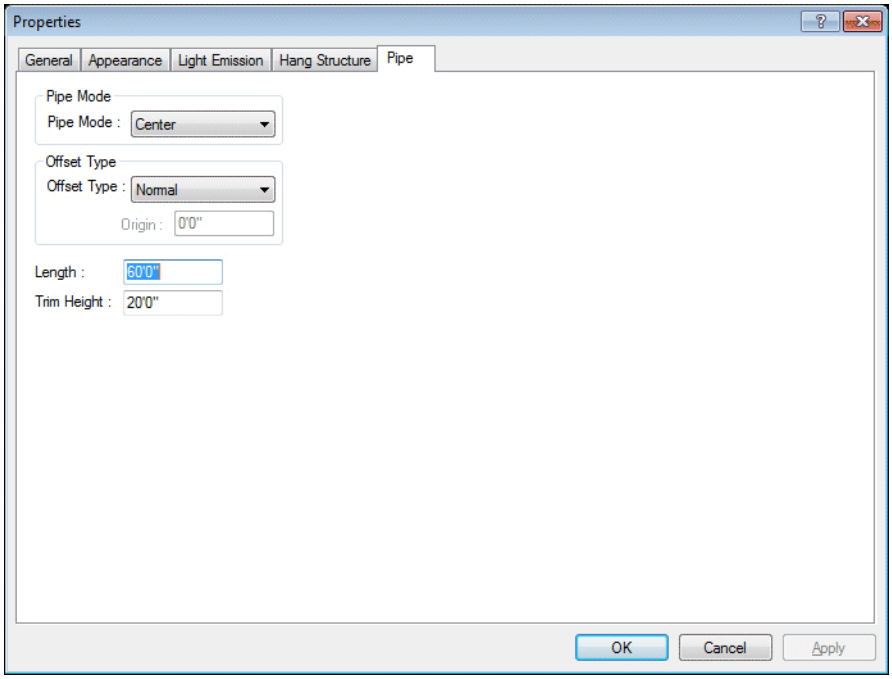
12 Click the *Hang Structure* tab.

13 From the Name drop-down list, select the name for this new pipe. If the name is not already in the list, use the *Position Manager* [...] to enter the new name.

14 Repeat to draw more pipes.

Pipe properties

Options on the *Pipe* tab affect the mode, offset, and dimensions of a pipe.



- *Pipe mode*: Select the pipe mode for the specific pipe. The available modes are:

- *Start*. If enabled, pipes are drawn from one end to the other. When start mode is used, offset is measured from the beginning of the pipe in a positive direction.
- *Centre*. If enabled, pipes are drawn from the center out to both ends at the same time. When center mode is used, offset values are either positive or negative, measured from the center of the pipe.

- *Offset Type*: Offset is a fixture property that measures where a fixture is hung on a pipe. The value depends on the pipe mode selected.

  Pipes can also base the Offset on a specified coordinate. In this case, a fixture's offset value is calculated as the distance on the specified axis from the specified coordinate. This is handy if you have many small pipes grouped together to form one hang position, for example the Balcony Rail. If this is the case, choose all the pipes' properties to have their Offset based about X=0. Fixtures on the Balcony Rail will then report where they are relative to the theatre, rather than the beginning of the pipe.

  This offset methodology can be extended to box booms as well. For example, a boom in the first box (which is 10' above the deck) can be drawn as a vertical pipe and have its offset to be based about Z=10'. A fixture's offset will then tell you how high (from the bottom of the box) to hang it.

  In WYSIWYG, set the offset mode that you want to use. The choices are:
  - *Normal*, which indicates that the offset is based on the pipe mode.
  - *X*, *Y,* and *Z,* which calculates the offset based on the specified coordinate.

  If you have selected an X, Y, or Z offset type, then you must specify a coordinate on the axis from which offset values will be calculated.
- *Length*: Type new text to change the length of the selected pipe.
- *Trim Height:* Type new text to change the trim height. If the pipe is vertical, this value determines the lowest Z value of the pipe.

Hanging truss

Truss is a hang structure with multiple hanging members. Truss differs from pipes in that it is built from library objects instead of being drawn. WYSIWYG contains an extensive library of different truss types from many different manufacturers. The truss library also contains floor mounts, booms, and ladders.

Indicators for truss assembly

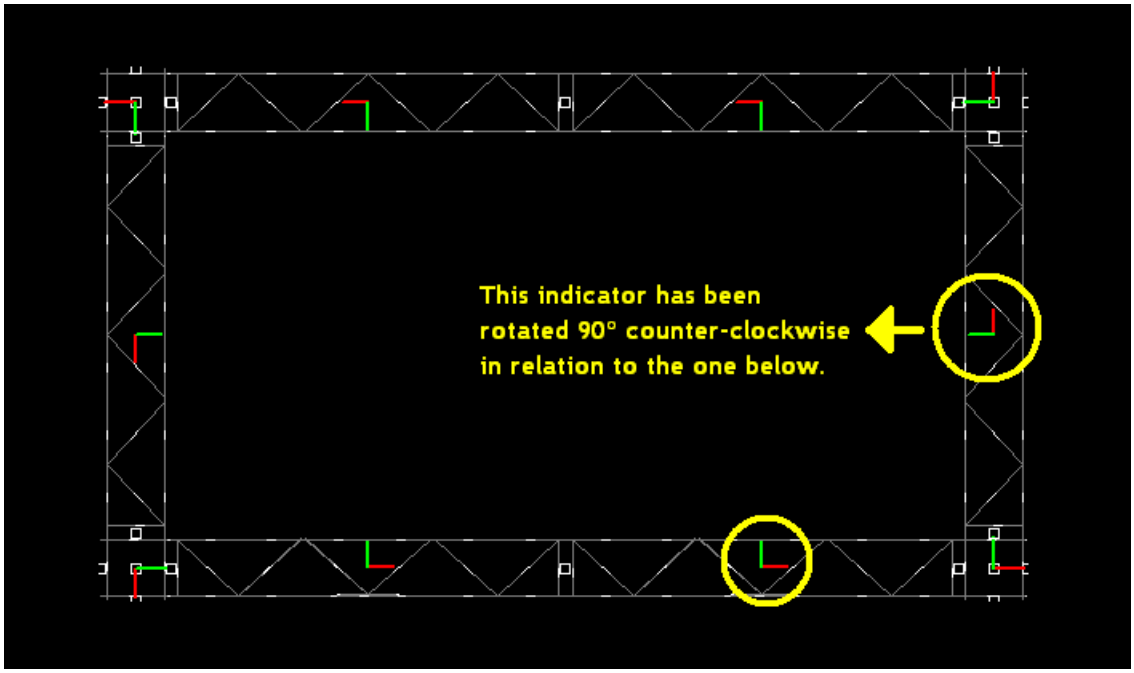
When hanging or selecting truss, a 3D indicator appears, displaying a directional/ positional reference for the individual piece of truss. You can use these indicators as a visual guide, helping you to assemble truss structures and instantly determine if a truss piece was not assembled as intended.

Tips:

- These 3D indicators are in no way related to WYSIWYG's coordinate system.
- It is highly recommended that you build truss structures in Isometric view and follow the left-to-right truss assembly rule.
- When a truss structure is completed, as long as the indicators for all the component pieces align (taking into consideration corner blocks and connectors – see example below), fixtures will hang correctly from this structure.

Indicators are visible whenever you select truss or when you are in the process of assembling/inserting it. When you are snapping a new piece of truss to an existing piece, if you see that the indicator for the existing piece does not align with the indicator for the new piece, right-click and select a different mount point, or roll the truss as necessary (rolling truss usually applies to triangular truss or with corner blocks), to ensure that the two indicators align.

Naturally, corner blocks and connectors have to be taken into consideration. For example, in the truss structure shown below (starting from the bottom left-most section and traversing counter-clockwise), the two truss pieces along the bottom have matching indicators.



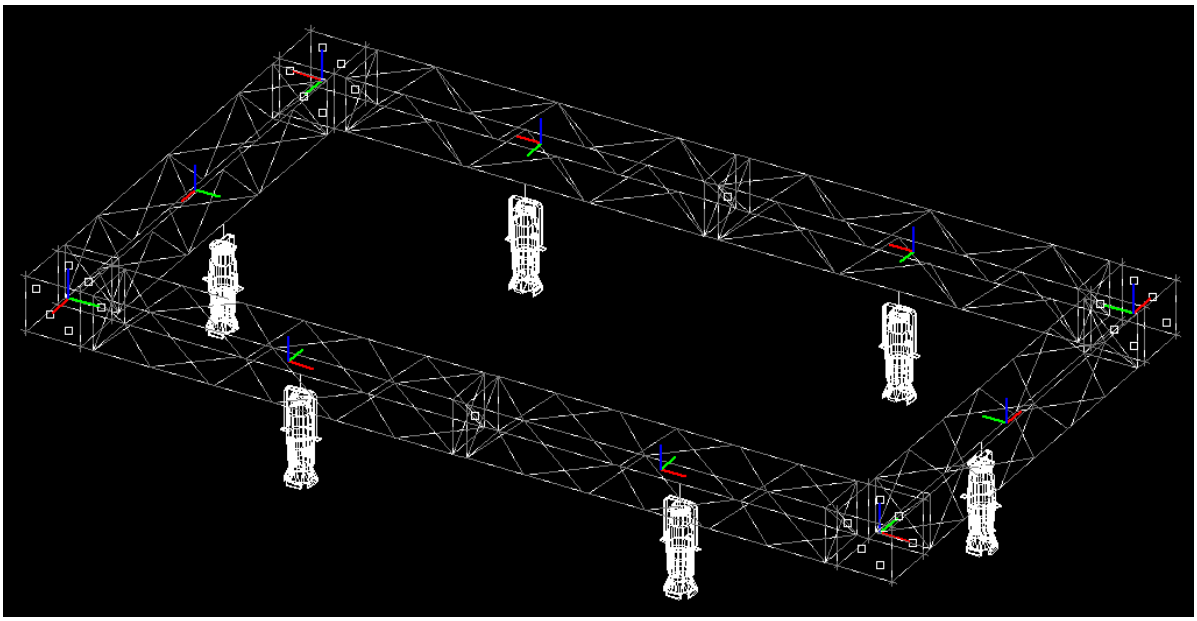
When the 90° corner block is used, the “running direction” of the truss changes. As such, the next truss piece is now rotated as shown by its indicator. This continues all the way around this rectangular truss structure until we connect back to the bottom left corner.

As the example demonstrates, while not all indicators will align in the same orientation (which would be impossible), the assembly may still be correct. When creating truss structures, take a moment at each corner block to analyze how the running direction of the truss changes due to the corner block. To do so, look at the indicator alone and visualize what would happen if you rotated it by 90° (or another angle, depending on what corner block/connector you are using).

If the indicator of the existing piece had the red line pointing to your right and green pointing up, rotating it 90° counter-clockwise would cause the green line to point to your left and the red to point up, while rotating the indicator 90° clockwise would cause the green line to point right and the red arrow to point down.

These are the only two cases in which fixtures hanging from the two pieces attached to the 90° corner block would behave “as expected”; should the indicators appear in any other positions, undo and assemble the truss again, making sure not to activate any of the right-click roll/alternate mountpoint options as you begin the assembly.

As you can see from the isometric screenshot of this same truss structure (below), all of the blue indicators are pointing in the same direction, which means that all lights hung from this truss will point downwards to the floor.



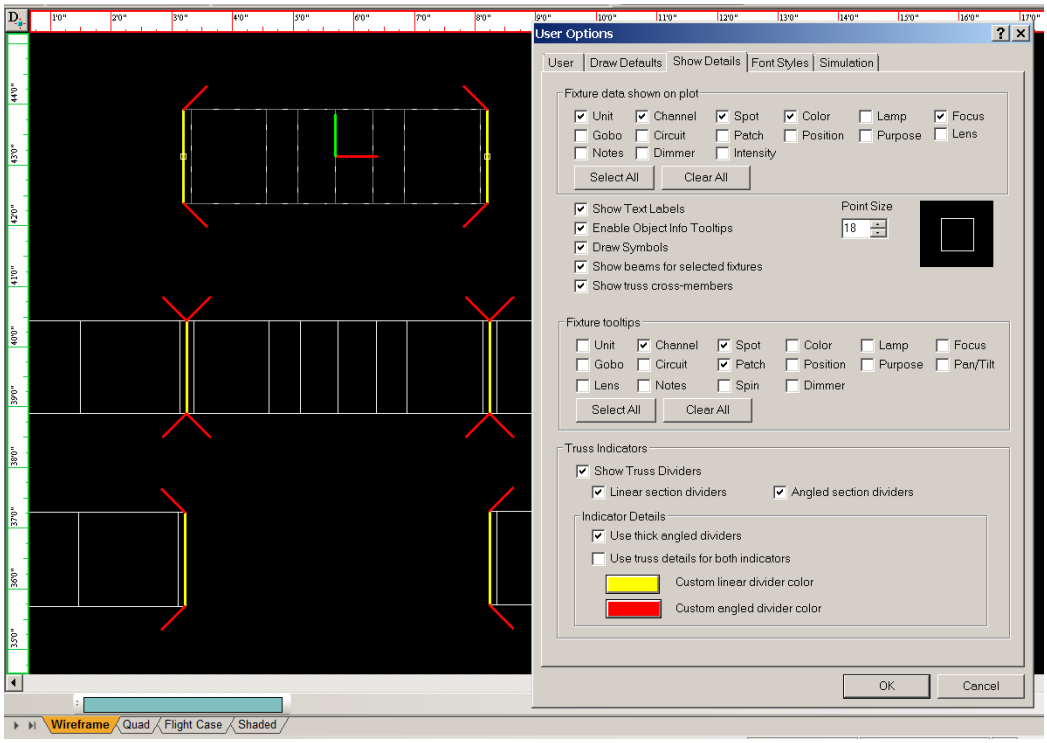
Configuring truss indicators

You can configure how the breaks in your truss rig appear in your drawing, a handy feature for showing your stage crew and production house quickly and simply the features of your truss structure.

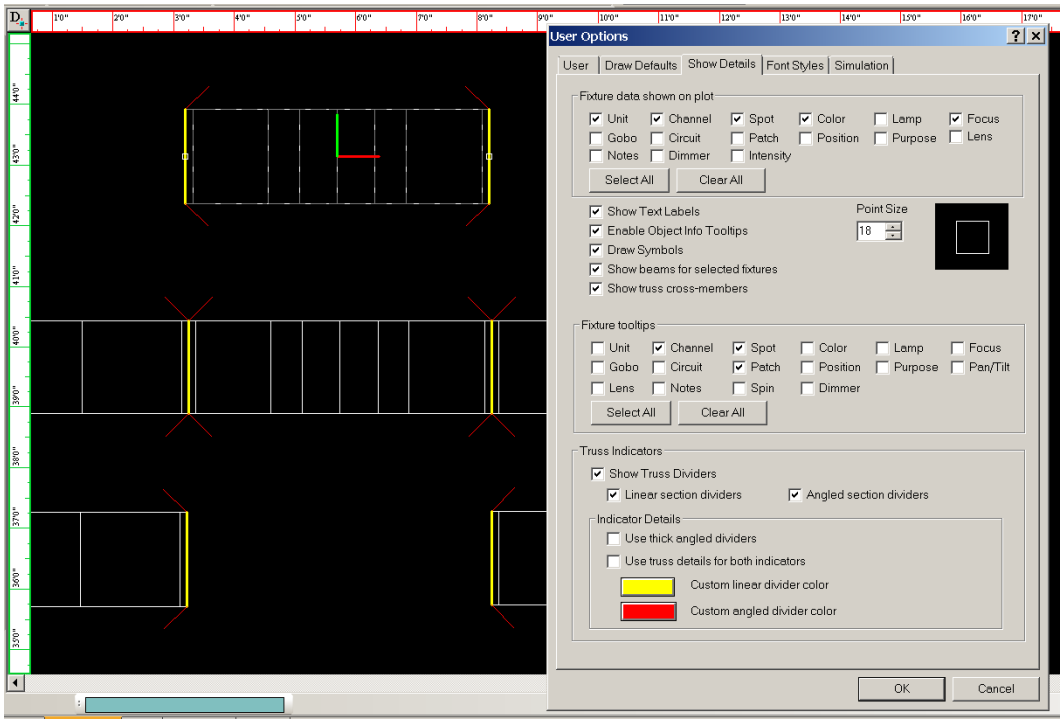
This feature is configured on the User Options > Show Details tab. For details, see “Show Details tab” on page 405.

For example, the following graphics show two files, the first with thick angled dividers, the second with thin angled dividers:

Truss with thick angled dividers



Truss with thin angled dividers



There are two types of Truss Indicators:

- *Linear Section Dividers* – Lines that appear parallel to the ends of your truss sections. These lines are always thick and can be colored any way you choose.
- *Angled Section Dividers* – Lines that appear at the junction of truss sections on a 45 degree angle with the truss end. The Angled Section dividers can be thick or thin, and can be colored any way you choose.

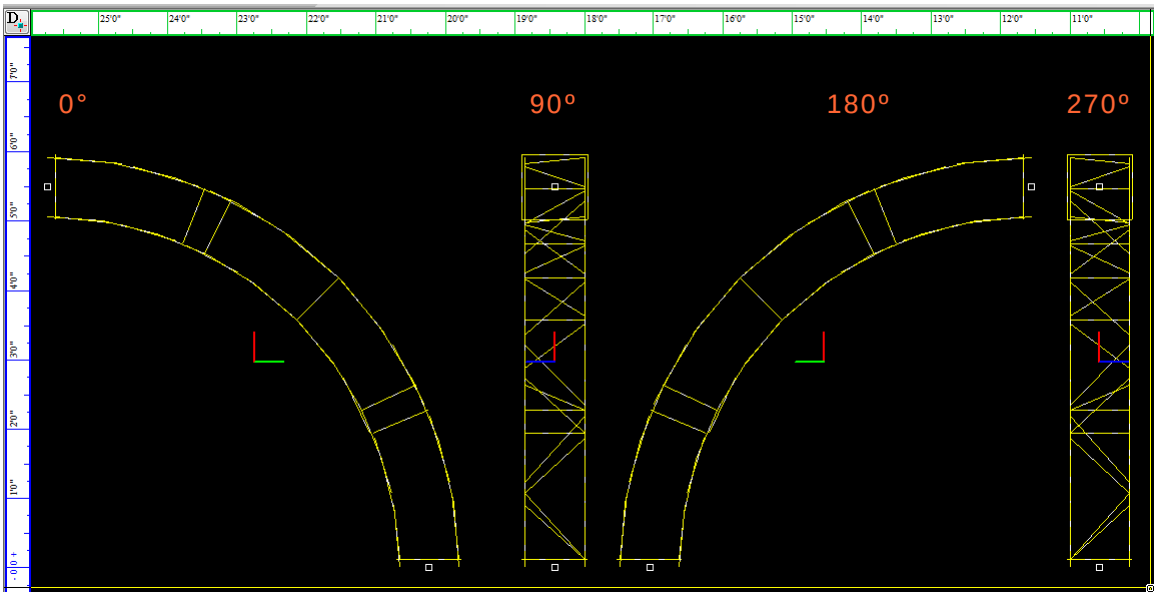
To configure truss indicators

1. Turn on truss indicators by clicking Options > User Options > Show Details.
2. Click *Show Truss Dividers*.
3. Click the boxes for Linear and Angled section dividers as you like.
4. You can choose to display thicker angled section dividers if you like by clicking the *Indicator Details* box below the option for on and off, and change colour for both linear and angled section dividers.

To hang truss

Before you hang truss, it is highly recommended that you enable Truss Assembly Snap, which connects truss pieces as you insert them, ensuring proper structural assembly and grouping. This is especially useful for long or complex structures made up of many individual pieces of truss. As with all snap functions (discussed in “Snaps” on page 105), you can select Assembly snap in mid-command or as a running snap type. For details, see “To select Assembly snap as a running snap type” on page 268 or “To select Assembly snap in mid-command” on page 268.

Note: Truss can now be hung horizontally or vertically. When a truss is hung vertically, the rotation of the truss is also set. The rotation angles available to vertical truss are 0°, 90°, 180°, 270°. The angles are based on the same Cartesian system used throughout WYSIWYG: 0º is the +X axis, 90º is the +Y axis, 180º is the -X axis and 270º is the -Y axis.

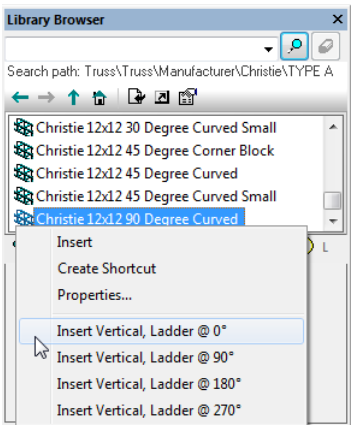


1 Open the Library Browser, and then click the *Truss* tab.

Tip: You can also click the *Truss* tool on the *Draw* toolbar to place truss. 

2 Navigate through the library to find the type of truss you want to use.

a. To insert the truss horizontally, double-click on the truss name. To insert the truss vertically, right click and select the rotation of the truss from the list.



Result: A piece of truss attaches to the cursor.

3 Click on the drawing to place the first piece of truss.

4 Continue placing pieces of truss as needed, using the truss indicators as guides when attaching one piece of truss to another. For more information, see “Indicators for truss assembly” on page 263. It is strongly recommended that you use Assembly snap when building truss structures. See “Using Assembly snap with truss” on page 268 for more details.

Note: To change the truss type, return to the library, double-click on the name of the new desired truss type and continue placing truss pieces as normal.

Tip: You can also use Truss shortcuts.

5 Right-click and choose **Finish Placing Truss**.

6 Right-click on the truss that you have just inserted, and then click **Properties**.

7 Click the *Hang Structure* tab.

8 From the Name drop-down list, select the name for this new truss. If the name is not already in the list, use the *Position Manager* [...] to enter the new name.

9 Click *OK*.

To insert a shortcut for truss objects

Refer to “To create a new shortcut” on page 56.

Using Assembly snap with truss

Assembly snap assembles truss pieces together as you insert them, ensuring proper structural assembly. This is especially useful for long or complex structures made up of many individual pieces of truss.

As with all snap functions (discussed in “Snaps” on page 105), you can select Assembly snap in mid-command or as a running snap type.

To select Assembly snap as a running snap type

Enable Assembly snap before inserting truss to assemble and group all subsequently inserted pieces of truss.

To select Assembly snap in mid-command

Enable Assembly snap after inserting the first piece of truss to assemble selectively. Assembly snap disengages after you place the next piece of truss, assembling only the first truss (or group of trusses) with the second piece.

To use Assembly snap

1 From the **Tools** menu, click **Snap**, and then choose **Assembly**.

Tip: You can also click the *Assembly* snap tool on the *CAD Options* toolbar. 

2 Insert the first piece of truss as described above. To snap truss pieces together, hover the cursor over the end of the piece of truss to which you want to attach the next piece and it will automatically snap into position. For added guidance, use the truss indicators to ensure that the truss is placed correctly. For more information, see “Indicators for truss assembly” on page 263.

3 Click to insert the piece of truss in the drawing.

Rolling truss and selecting alternate mount points during assembly

When inserting truss pieces in your drawing, you can roll them into the desired position by using the Roll tool in the right-click menu or the Roll hotkeys (X and Z). By enabling Assembly snap, pieces of truss that you drag into the drawing and bring close to the end of an already-inserted piece of truss will snap to connect to that truss, ensuring proper structural assembly.

Notes:

- By default, truss will always insert parallel to the X-axis. If the truss structure you are building needs to start with a piece in a different position, you must insert that piece, right-click to finish inserting it, and then use WYSIWYG's Rotate function (**Edit** > **Rotate**), to adjust it to the desired position. The *Truss Roll* commands can only be used when subsequent pieces of truss are inserted.
- When rolling truss, use the truss indicators as guides. For more information, see “Indicators for truss assembly” on page 263.

Rolling truss

It is sometimes necessary to roll truss—most often when you use corner blocks that change the (vertical) running direction of the truss.

The Truss Roll commands allow truss to be "rolled" or "spun" around its long axis, and are only available when you are attaching truss to an existing piece of truss (in the right-click menu and/or via hotkeys).

Note: You should only roll truss when you notice that the cords of the piece that you are attempting to assemble do not connect properly to the existing piece.

To access the Roll options, you must have already inserted a piece of truss and the next piece must be attached to your cursor and ready to be attached to an existing piece. When you right-click, a menu appears that allows you to change the roll of the truss in 90º increments.

Tips:

In addition to, or instead of, the right-click menu, use the Z and X keys to cycle through the standard Roll options:

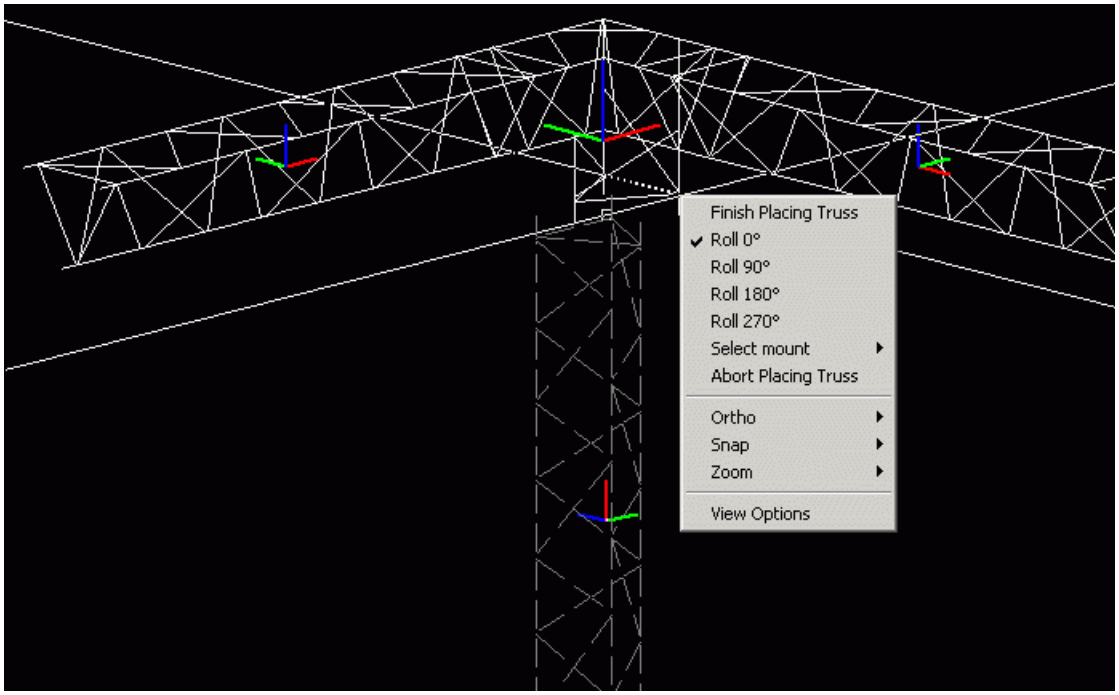
- Pressing Z will cycle forward through the options. For example, if starting at the default Roll 0º, pressing Z once will change to the Roll 90º, pressing it again will change to Roll 180º, and then to Roll 270º, and finally back to Roll 0º.
- Pressing X will cycle backward through the options. For example, if starting at the default Roll 0º, pressing X once will change to the Roll 270º, pressing it again will change to Roll 180º, and then to Roll 90º, and finally back to Roll 0º.

The selection of a Roll option is NOT maintained after the piece of truss that it was used on, was assembled. In other words, once a piece of truss was assembled into the structure, the Roll option will ALWAYS change back to Roll 0º. It is not possible nor desired to maintain the Roll option since there are virtually no cases in which this would be required.

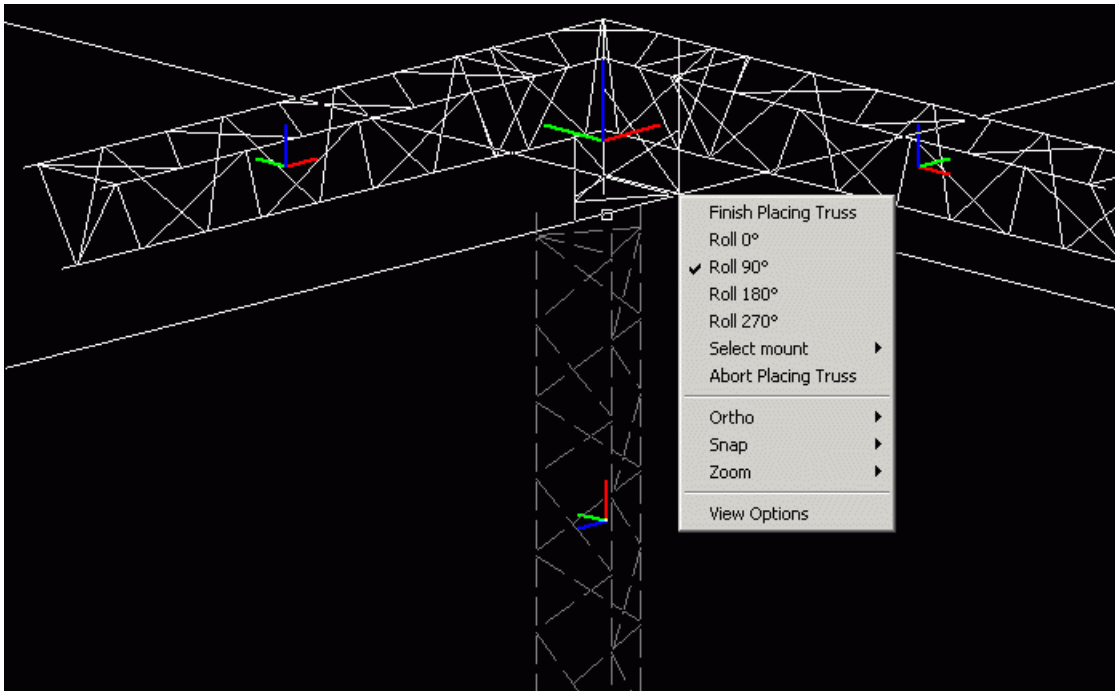
Examples

In the images below, you can clearly see that the cords of the piece that's about to be attached to the 'down' leg of the corner block do not align/connect properly with the default "Roll 0º" position, so a different Roll option must be selected. After cycling through the four Roll options (0º, 90º, 180º and 270º), it is clear that the Roll 270º is the correct position.

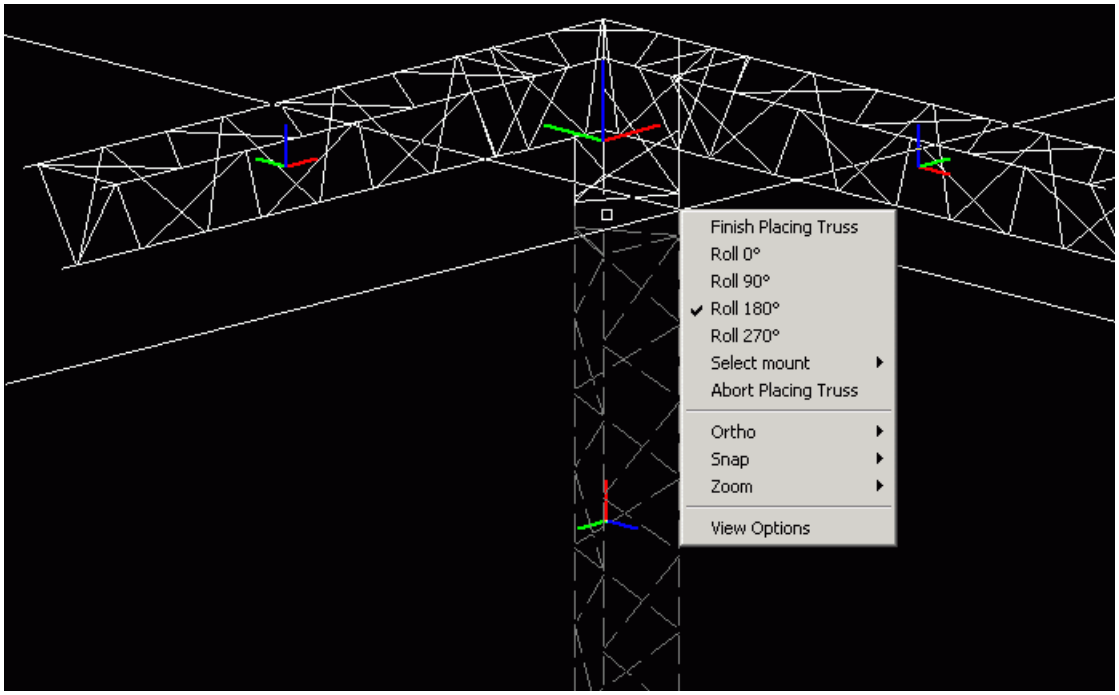
Roll 0°



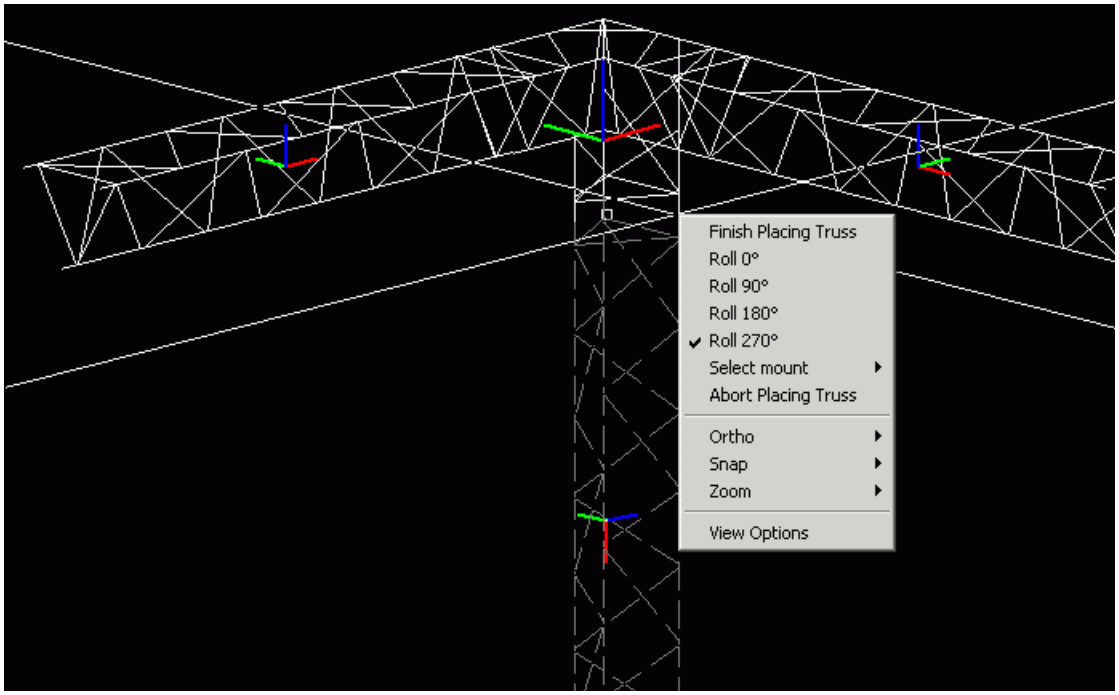
Roll 90°



Roll 180°



Roll 270°



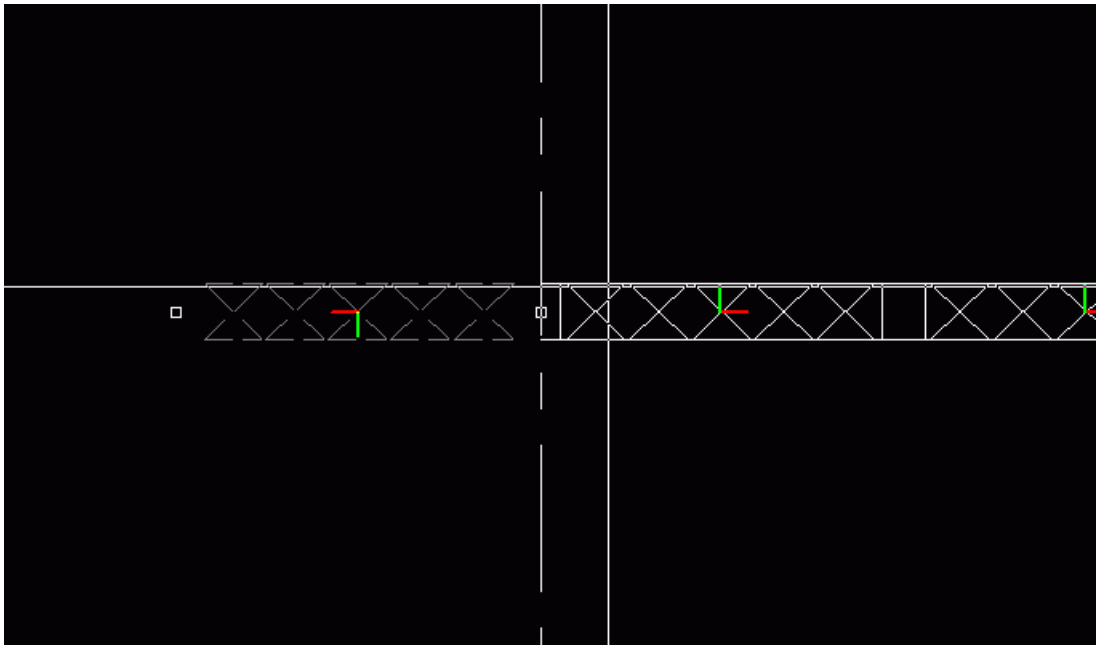
Select Mountpoint options

A Mountpoint is any part of a truss section, corner block and/or connector to which another section, corner block or connector can be attached.

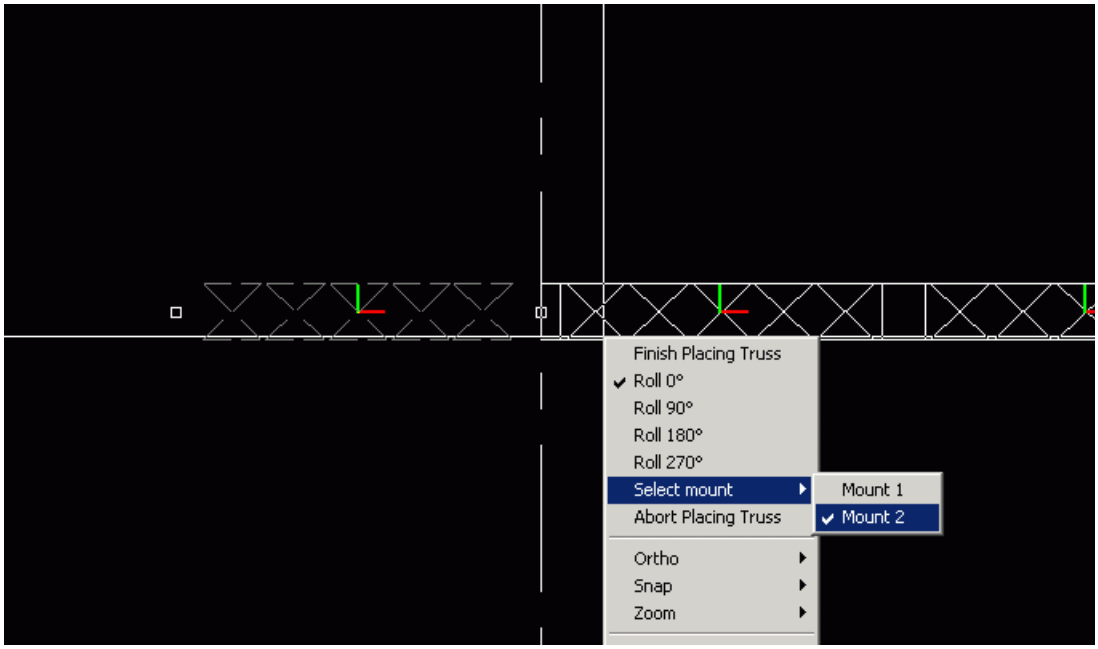
As with the Roll options, the Select Mountpoint options are only available (in the right-click menu and/or via hotkey) while truss is being attached to an existing piece of truss and the next piece/section, corner block and/or connector must be on the cursor and ready to be attached to the previous pieces. Their purpose is to allow a different Mountpoint of the truss to be selected during assembly.

Selecting a different Mountpoint is *sometimes* necessary when you are attaching a connector or corner block to a straight or curved section of truss, in case attaching with the default Mountpoint is not desired or would result in an incorrect structure.

Selecting a different Mountpoint is *always* necessary when you are assembling straight or curved truss section runs from the center out. For example, if you were to build the Front of House truss starting at the centreline and first attaching sections towards the right (stage left in some cases), and then attaching sections from the center towards the left (stage right in some cases), you would need to change to Mount 2 once you start to attach sections towards the left; otherwise, the Truss Indicators would show you that the running direction of the truss is no longer correct, as shown in the graphics below.



Here, the section left of center has been attached without selecting the Mount 2 Mountpoint. Note that the Truss Indicator for this section appears in the opposite direction from the Truss Indicators for the pieces right of center.



When you right-click and select the *Mount 2* Mountpoint before attaching the truss, the Truss Indicator for the truss section left of center matches the Indicators for the existing sections.

Notes:

- In addition to, or instead of, the right-click menu, you may use the S key to cycle through the available Mountpoints. Remember that for many truss pieces (and especially for sections), their appearance may not change when a different Mountpoint is selected, but the Truss Indicator will always change.
- The selected Mountpoint is maintained until a new piece of truss is selected for insertion. This is to facilitate building truss structures in the fashion described above (i.e., starting at the center and building towards the right, and then coming back to center and building towards the left).

Hanging and focusing fixtures

Introduction

Fixtures are objects on the plot and appear in 3D in shaded views. When you insert a fixture into the plot, automated fixtures can be hung from a hang structure or inserted on the floor. Manual fixtures must be hung from a hang structure. WYSIWYG has an extensive library of fixtures to choose from.

Inserting fixtures into your drawing is as easy as pointing and clicking. You can insert individual fixtures directly from the Library Browser, or you can create shortcuts to your favorite fixtures.

To quickly hang multiple fixtures on a pipe or straight truss simultaneously, you can use the array feature; to arrange multiple fixtures along a pipe/truss, use the distribute tool. For details, see “Hanging multiple fixtures” on page 275 and “Distributing fixtures” on page 276.

Inserting fixtures

To hang fixtures

Use this procedure to hang fixtures one by one on lengths of pipe or truss. To hang multiple fixtures simultaneously, use the Array feature. For details, see “To hang multiple fixtures” on page 275. After hanging the fixtures, you can arrange them simultaneously on pipes/truss by using the distribute feature. For details, see “Distributing fixtures” on page 276.

1. From the **Library** menu, choose **Browse Library.** Alternately, click the Fixture icon on the **Draw** toolbar.
2. Click the *Fixture* tool at the bottom of the browser. 
3. In the menu that is now visible, double-click *Manufacturer* to select fixtures by manufacturer name, *Type* to select by fixture type, or *All* to see all the fixtures in alphabetical order.

   Note: Alternately, you can use the Library Browser’s Search function to locate the fixture that you want to insert. Simply type the name of the fixture (or a partial name) in the Search box at the top of the Library Browser, and then click the Search icon. For details, see “To search the Library Browser” on page 134.
4. Double-click on the name of the fixture that you want to insert.

   Result: A fixture with default settings for this type attaches to the cursor.

   Note: You can change the fixture frame size using the Add Accessory option. For more information on adding accessories, refer to “To insert an accessory” on page 315.
5. To change the properties of the fixture before inserting, right-click on the fixture name, and then click **Properties**.

   Result: A dialog box opens with a shaded view of the fixture on the left and an image of its symbol on the right. Use the *Photometrics* tab to change the lamp and lens settings for the fixture, if applicable. For more information on the Photometrics tab, refer to “Photometrics tab” on page 299. Click *Insert* and the dialog will close.

6 Move the mouse over a hang structure, and then click to place the fixture. Continue placing this type of fixture by clicking on the other hang structures as desired.

7 Finish placing this fixture type by right-clicking, and then choosing **Finish placing fixtures** from the resulting menu.

8 To place other fixture types, repeat the above steps for each type.

To insert fixtures on the floor

Use this procedure to insert automated fixtures one by one on the floor, without first attaching them to a pipe or truss. Fixtures inserted this way support CAD operations e.g., copy/paste, move, and rotate.

Note: Only automated fixtures can be inserted on the floor.

1 From the **Library** menu, choose **Browse Library.** Alternately, click the Fixture icon on the **Draw** toolbar.

2 Click the *Fixture* tool at the bottom of the browser. 

3 In the menu that is now visible, double-click *Manufacturer* to select fixtures by manufacturer name, *Type* to select by fixture type, or *All* to see all the fixtures in alphabetical order.

Note: Alternately, you can use the Library Browser's Search function to locate the fixture that you want to insert. Simply type the name of the fixture (or a partial name) in the Search box at the top of the Library Browser, and then click the Search icon. For details, see "To search the Library Browser" on page 134.

4 Right-click on the name of the fixture that you want to insert. In the menu that appears, select **Insert on Floor**.

5 Move the mouse anywhere in the plot, and then click to place the fixture. Continue placing this type of fixture by clicking as desired.

6 Finish placing this fixture type by right-clicking, and then choosing **Finish placing fixtures** from the resulting menu.

7 To place other fixture types, repeat the above steps for each type.

Hanging multiple fixtures

Once you hang a single fixture, you can use the Array feature to instantly copy and paste multiple copies of it along the length of a pipe or straight piece of truss in the direction that you specify. This feature enables you to hang up to 100 fixtures quickly and precisely.

To hang multiple fixtures

1 On a pipe or a straight length of truss, hang the fixture of your choice by following the procedure "To hang fixtures" on page 274.

Note: To hang fixtures along the length of truss or pipe, hang the first fixture at one end of the pipe/truss and then set the direction in which you want to copy and paste it.

2 Click the fixture to select it, and then right-click and choose **Tools** > **Array** > **Along Pipe**.

Result: An arrow attaches to your cursor when you hover over the pipe/truss.

3 Click on the pipe/truss to set the direction in which the fixture will be arrayed, either to the left or right of the fixture.

4 In the dialog box that opens, type the total number of copies desired (this includes the original object). Adjust the interval value, if required.

5 Click *OK*.

Result: The fixtures are placed along the pipe/truss.

Selecting fixtures

You can use the Select feature to select all the fixtures in your plot, only certain types of fixtures, or only the fixtures on a certain hang structure. These features can also be accessed from the Fixture Selection toolbar.

This tool is most useful when all fixtures of one type need to be replaced with fixtures of another type, as well as when exporting fixture point-of-view (PoV) images to Luminair. For details on Luminair, see “Connecting to Luminair” on page 581.

To select all fixtures

1 In your plot, right-click and choose **Select Fixtures > Select All Fixtures**.

Result: All fixtures are selected.

To select all fixtures of a particular type

1 Click to select the fixture that you want to select.

2 Right-click and choose **Select Fixtures > <CHOOSE FIXTURE NAME>**.

Result: All fixtures of that type are selected.

To select a certain type of fixture on a particular type of hang structure

1 Click to select the hang structure containing the fixture that you want to select.

2 Right-click and choose **Select Fixtures > Select on Selected Pipe/Truss** and then choose the type of fixture that you want to select.

Result: All fixtures of the specified type are selected on the chosen type of hang structure.

To select all fixtures on a particular hang structure

1 Click to select the hang structure containing the fixtures that you want to select.

2 Right-click and choose **Select Fixtures > Select on Selected Pipe/Truss > Select All**.

Result: All fixtures are selected on the chosen type of hang structure.

Distributing fixtures

You can use the Distribute feature to quickly arrange multiple fixtures evenly over a pipe or piece of straight truss starting at a specific point.

You can distribute selected fixtures in three ways:

- between two points that you choose on the pipe/truss
- in a specific direction from one point that you choose on the pipe/truss
- centered around a specific point that you choose on the pipe/truss

To distribute fixtures between two points

Note: Since you cannot mount fixtures directly on the end point of truss, before you distribute fixtures between two points, ensure that End Point snap is disabled.

Use this procedure to select two points on a pipe or length of straight truss and have WYSIWYG distribute selected fixtures evenly between those points. The spacing is determined by dividing the selected region of the pipe/truss by the number of fixtures selected.

1. Click to select the fixtures that you want to distribute on the pipe/truss (to select multiple fixtures, press Ctrl and then click on each one).
2. From the **Tools** menu click **Distribute** > **Between two points**.
3. Click to set the two points on the pipe/truss.

   Result: The selected lights are evenly spaced between the two points.

To distribute fixtures in a direction

Use this procedure to select a point on a pipe or length of straight truss and have WYSIWYG distribute selected fixtures along the remaining length of the pipe/truss in the direction that you choose, spaced according to your specifications.

1. Click to select the fixtures that you want to distribute on the pipe/truss (to select multiple fixtures, press Ctrl and then click on each one).
2. From the **Tools** menu click **Distribute** > **In a direction**.
3. Click to set the starting point on the pipe/truss from which you want to distribute the selected fixtures.
4. Click on one side of the point (either to the left or right of it) to choose the direction in which the fixtures will be distributed.

   Result: The *Distribute - In a direction* window appears.
5. Type the desired distance between each fixture, and then press *OK*.

   Note: The value that you type must allow the selected number of fixtures to be evenly distributed over the remaining length of the pipe/truss.

   Result: The selected lights are evenly spaced between the two points.

To distribute fixtures around a central point

In this procedure you choose a point on the pipe/truss and then distribute selected fixtures along the pipe/truss, using this point as the center. You specify the spacing of the fixtures.

1. Click to select the fixtures that you want to distribute on the pipe/truss (to select multiple fixtures, press Ctrl and then click on each one).
2. From the **Tools** menu click **Distribute** > **Center**.
3. Click on the pipe/truss to set the point around which the selected fixtures will be centered.

   Result: The *Distribute - Center* window appears.
4. Type the desired distance between each fixture, and then press *OK*.

   Note: The value that you type must allow the selected number of fixtures to be evenly distributed over the length of the pipe/truss around the point that you have specified.

   Result: The selected lights are evenly spaced around the point.

Replacing fixtures

When the fixture requirements for a show change, instead of deleting and re-hanging fixtures, you can use the Replace Fixture function to quickly and easily replace one or more fixtures with another.

To replace fixtures

1 Select the fixtures that you want to replace using one of the selection methods described in “Selecting fixtures” on page 276.

2 Right-click and choose **Replace Fixture**.

3 Select the new type of fixture from the Fixture Selection dialog box.

4 Click *Insert*.

Result: All the fixtures that you selected in step 1 are replaced with the fixture type chosen in step 3. Some fixture attributes may be modified as a result of the replacement.

Custom fixtures

To create a custom fixture

You must create custom fixtures on the WYSIWYG welcome screen with all files closed. If you are currently working on a document, save and close it.

Note: You do not have to complete all fields. Fill in as much information as you can. You cannot edit custom fixtures once saved, but you can delete them and create them from scratch with new properties.

1 On the WYSIWYG welcome screen, from the **Options** menu, choose **Create new conventional**.

Result: The *Fixture Wizard* opens.

2 Select the type of fixture that you want to create.

3 In the *Name* box, type a name for the new fixture.

4 Click *Next*.

5 From the *Type* drop-down list, select the circuit type. The default, REGDIM, is a regular dimmer.

6 Click *Next*.

7 Add or modify the path.

Note: The path is the location in the library where the fixture will be stored. A fixture assigned to the path “Custom/Fresnels” would be available in the custom section of the Fixture library in a sub-category named FRESNELS.

8 Click *Next*.

9 Select the lamp base and default lamp for this fixture.

10 Click *Next*.

11 Specify the lens system for the fixture.

12 Click *Next*.

13 Type a value in degrees for the beam angle for this fixture.

14 Type a value in degrees for the field angle for this fixture.

15 Click *Next*.

16 Select the color frame slot size.

17 Click *Next*.

18 From the symbol library, select a 2D symbol for this fixture.

19 Click *Next*.

20 From the shape library, select a 3D symbol for this fixture.

Note: A 10” parcan is shown beside the selected shape in the viewing pane to offer a relative size measurement.

21 Click *Next*.

22 Type the manufacturer, weight, and any other information.

23 Click *Next*.

24 Type your name

25 Click *Finish*.

To delete custom fixtures

If you do not want to keep a custom fixture any longer, you can delete it from the library browser.

Notes:

- You cannot delete a custom fixture if it is currently inserted in your plot; instead, you must delete the item from your plot, save and close the file. When you reopen the file you can delete the item from the library browser.
- When you delete a custom fixture, you do not delete the shortcut associated with it. You must delete shortcuts manually.

1 In the Library Browser, navigate to the folder where you stored the custom fixture.

2 Highlight the fixture name, and then right-click and choose **Delete**.

Note: This option is disabled if the item is currently inserted in your plot. You must delete the item from your plot, and then save and close the file. When you reopen the file you can delete the item from the library browser.

3 A message box appears telling you the action is irreversible. Click *Yes* to delete the fixture.

Pipe snap

Pipe snap places fixtures on a pipe at a specific interval. Pipe snap does not work for truss or other hang structures. It allows you to specify a pre-determined distance between fixtures.

As with all snap functions (refer to “Snaps” on page 105 for more information about snap tools), you can select Pipe snap in mid-command or as a running snap type.

To select Pipe snap as a running snap type

Enable Pipe snap before inserting fixtures to hang fixtures equidistantly.

To select Pipe snap mid-command

Enable Pipe snap after inserting the first fixture to enforce the interval selectively. Pipe snap will disengage after you place the next fixture.

To use pipe snap

1 From the **Tools** menu, choose **Snap**, and then choose **Pipe Snap**.

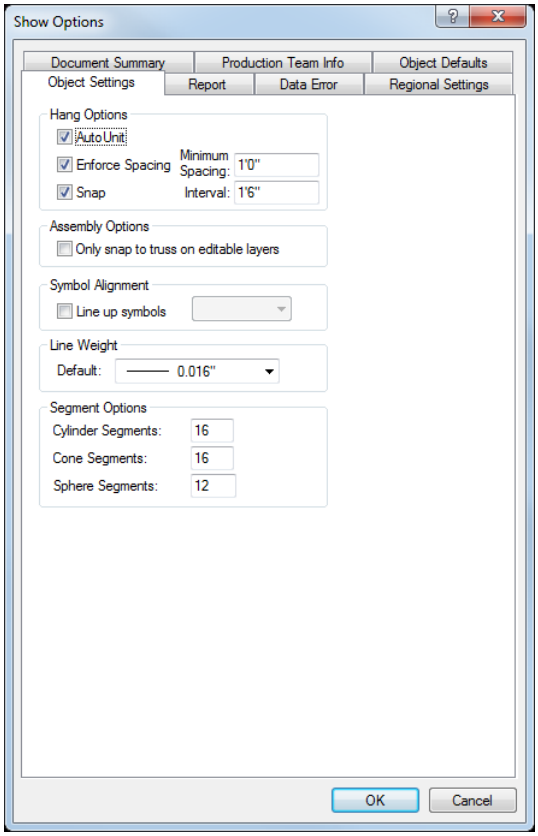
Tip: You can also click the *Pipe snap* tool on the *CAD Options* toolbar. 

2 Insert fixtures as described above. With pipe snap active, the fixtures are automatically hung at the pre-defined interval.

To set the snap interval

1 From the **Options** menu, choose **Show Options**.

2 Click the *Object Settings* tab.



3 Select the *Snap* check box if it is unchecked, and then type the new value in the *Interval* box.

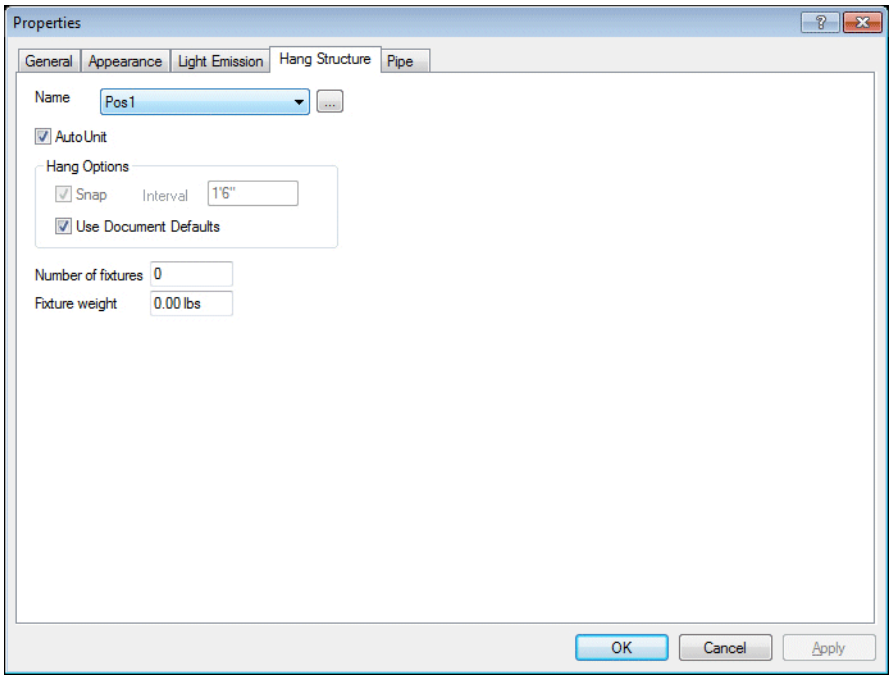
4 Click *OK*.

To set an interval for a specific pipe

1 Right-click the pipe.

2 Click **Properties**.

3 Click the *Hang Structure* tab.



4 Click to clear the *Use Document Defaults* check box.

5 Select the *Snap* check box.

6 In the *Interval* box, type a new distance. This is the measurement between fixture insertion points.

7 Click *OK*.

Focusing fixtures

You can focus the fixtures in your plot in multiple ways:

- You can click and drag the beam of a selected fixture.
- You can type the pan, tilt, and spin values for the fixture in its *Properties* box.
- You can assign a focus position or focus object to the fixture.
- You can use the Fanning Tool feature to focus conventional fixtures in CAD mode (you can use this feature to focus automated lighting in DESIGN mode)

To focus using click and drag

1 Ensure that the *Enable Beam Toggle* option is enabled on the CAD Options toolbar. 

2 Select the fixture that you want to focus.

Result: The fixture's beam appears.

3 Click on the beam and drag it to its new position. When you are done, release the mouse button.

Tips:

- You can also right-click the fixture, and then click **Focus** to drag the beam without having to hold the mouse button down continuously. Click to finish when the beam is focused where you want it.
- Use the missing focus coordinate to set a value on the inactive axis.

Drawing focus positions

Focus positions are

- points at which fixtures can be focused.
- objects in the drawing.

To insert a focus position

1 From the **Draw** menu, choose **Focus Position**.

Tip: You can also click the *Focus Position* tool on the *Draw* toolbar to place focus positions. 

2 Type a label for the focus position.

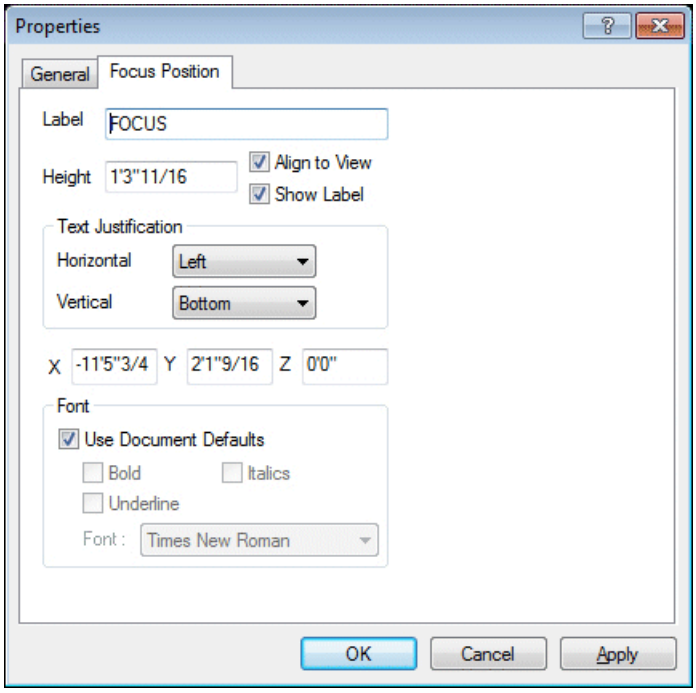
3 Move the mouse so that the intersection of the crosshairs is placed where you want the focus position.

4 Click to insert the focus position.

5 Repeat these steps to insert all focus positions.

Focus position properties

Options on the *Focus Position* tab affect the alignment and style of the selected focus position.



You can modify the justification, height, and presentation of a focus position. For a description of the justification and presentation-type properties, refer to the “Text label tab” on page 207.

Focus positions are unique in that you can adjust the focus position coordinates without actually moving the focus position on the drawing. The X,Y, and Z coordinate is the coordinate of the location in 3D space where the focus position is to lie.

Focus Objects

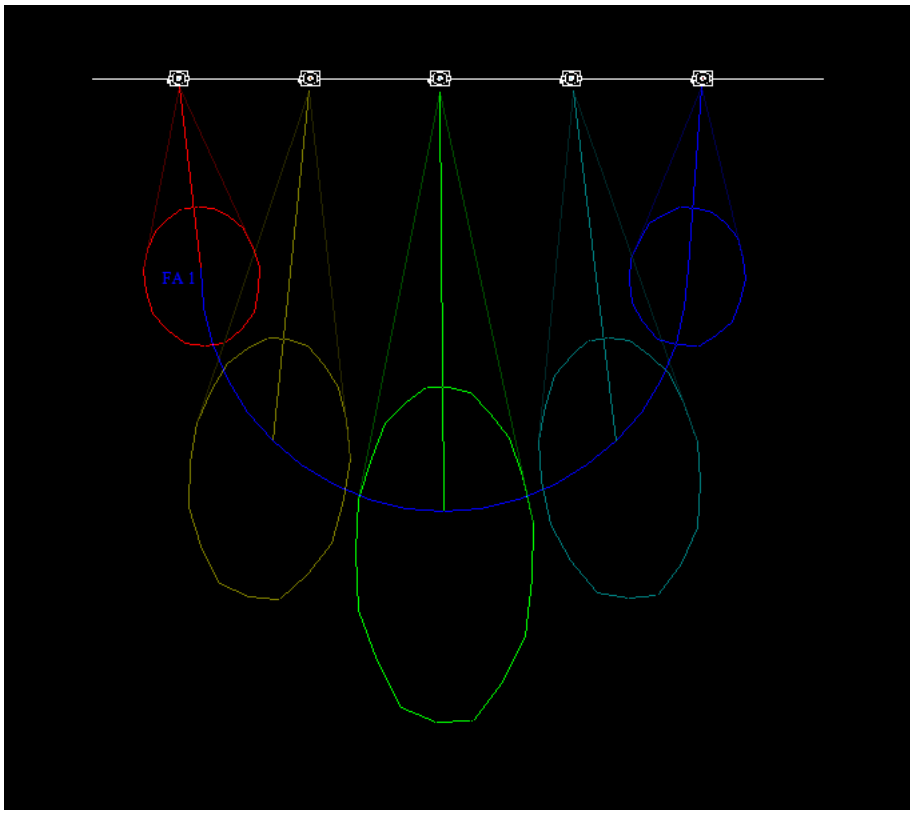
Focus Objects allow you to draw an object, either a line or an arc, like you would a Focus Point. Selected fixtures can then be assigned directly to the drawn object. If you move or resize the object, the fixtures assigned to the object move along with your Focus Object. Focus Objects have many of the same properties of a Focus Point, but have considerably more functionality.

Note: Conventional fixtures can be assigned to Focus Objects in CAD mode; moving head units can be assigned to those same Focus Objects in DESIGN mode.

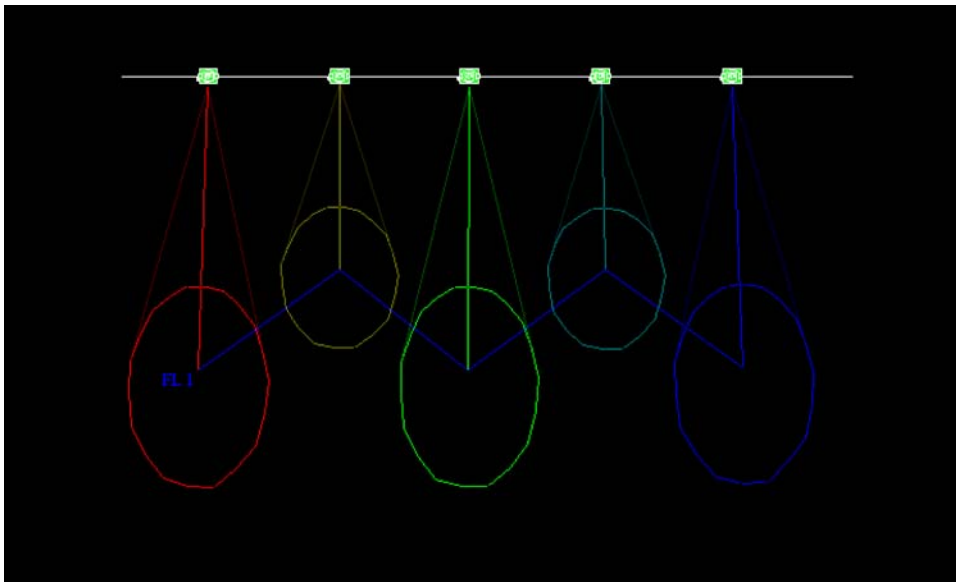
Linking fixtures to Focus objects

When you draw a Focus Object, you can assign several pairs of fixtures to that object as you would like, and the fixtures will be organized in one of two ways, either using *Spread Focus* or *Focus to Vertices*.

The *Spread Focus* function available is available to both focus lines and focus arcs. *Spread Focus* is where the object divides itself up based on the fixture “sets” that you select, and distributes the fixtures at even intervals along the object in the order selected.



Focus to Vertices is only available to focus lines. When using *Focus to Vertices,* the focus line is drawn with multiple segments separated by a vertices. When fixtures are assigned to the focus line, a single fixture will be focus to each vertex in the line in the order selected. The maximum number of fixtures that can be assigned is limited to the number of vertices in the focus line.



Breaking the fixture links to Focus Objects

Once a group of fixtures is assigned to a Focus Object, they are essentially “linked” to that Focus Object until otherwise refocused. If you have a group of fixtures assigned to a Focus Object and you refocus or move one of them, any of the other fixtures will remain in the position they were focused, but their link will be removed from the group of fixtures and the Focus Object, as you would expect.

Focus Objects - Point naming

Keep in mind that since you can assign different sets of fixtures to the same object, the same point can be used for two different fixtures. In Spreadsheets and Reports (and CAD), the position points in the Focus column are indicated as "<name> - 1 of 3", "<name> - 4 of 5," and so on (where one set has 3 fixtures and the other set has 5 fixtures). In this case, “1 of 3" and “1 of 5" would be the same point, as would “2 of 3" and “3 of 5", as would “3 of 3" and “5 of 5".

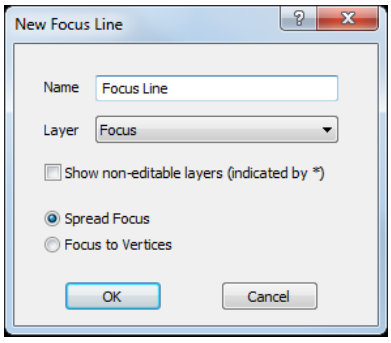
Assigning fixtures to Focus Objects

Creating Focus Objects and assigning fixtures to them is as easy as selecting the fixtures, selecting the Focus Object, and applying a key shortcut.

To assign conventional fixtures to a Focus Line in CAD mode

1 Draw a Focus Line by clicking **Draw** > **Focus Objects > Focus Line**.

Result: The *New Focus Line* window appears.



2 Type a name for the new focus line.

3 Select the layer on which you want to draw the focus line.

4 Select the focus type.

5 Click *OK*.

6 Draw your focus line where you like.

7 Select your fixtures in the order in which you would like to focus them.

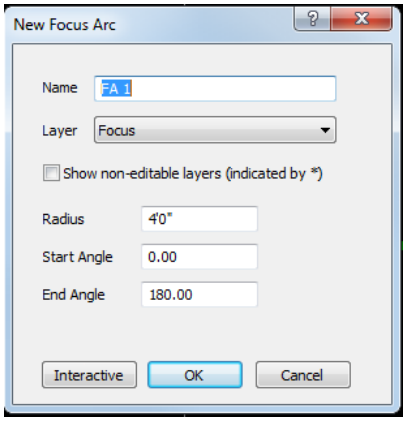
8 While keeping the fixture selection you just created, also select the focus line to which you want to assign conventional fixtures in CAD mode.

9 Use the keyboard shortcut CTRL+SHIFT+F to assign the fixtures to the focus line.

To assign conventional fixtures to a Focus Arc in CAD mode

1 Draw a Focus Line by clicking **Draw** > **Focus Objects > Focus Arc**.

Result: The *New Focus Arc* window appears.



2 Type a name for the new focus arc.

3 Select the layer on which you want to draw the focus arc.

4 To drawn the focus arc free hand, click *Interactive*. To pre-construct the focus arc, enter in the radius, start angle and end angle details, then click *OK*.

5 Place your focus arc where you like.

6 Select your fixtures in the order in which you would like to focus them.

7 While keeping the fixture selection you just created, also select the focus arc to which you want to assign conventional fixtures in CAD mode.

8 Use the keyboard shortcut CTRL+SHIFT+F to assign the fixtures to the focus arc.

Quick focus

Quick focus assigns a focus position or Pan and Tilt angle to the selected fixture.

To assign a focus position

1 From the **Tools** menu, choose **Quick Focus**.

Tip: You can also click the *Quick Focus* tool on the *Tools* toolbar. 

2 Click on the desired focus position. This will set that position as the destination for the next fixtures you select.

3 Click on each fixture that you want focused to that position or focus object.

4 Right-click anywhere in the wireframe view, and then choose **Finish Quick Focus**.

To assign a pan and tilt angle

Note: For more information on Quick Tools, see “Quick Fixture tools” on page 299.

1. From the **Tools** menu, choose **Quick Tools**.
2. In the dialog box that opens, check *Focus*.
3. Type values in degrees for *Pan* and *Tilt*.
4. Click *OK*.
5. Click on each fixture you want oriented to the specified angle.
6. Right-click anywhere in the wireframe view, and then choose **Finish Quick Tools**.

To focus from the fixture’s properties

1. Select the fixtures that you want to focus.
2. Right-click and choose **Properties**.
3. Click the *Fixture* tab.
4. Type values in degrees for *Pan*, *Tilt*, and *Spin*, or select a focus position from the drop-down list.

   Note: For the definitions of these attributes, please refer to “Data fields” on page 358.
5. Click *OK*.

Using the Fanning Tool feature in CAD mode

The Fanning Tool makes your aerial focuses and any other fan-like focus quick and easy, enabling you to select a group of fixtures, tilt and pan them as a group, and then spread the focus out with a simple wheel. This action can be performed in CAD mode (for conventionals) and Design mode (for moving heads).

To focus conventional fixtures using the Fanning Tool

1 Open the Fanning Tool with the toolbar button F .

Result: The Fanning Tool window appears.



2 Select a group of fixtures by drag-selecting or control-clicking in either Wireframe or Shaded View (in DESIGN mode, this is part of the Focus Designer tool).

3 Use the Pan/Tilt wheel to move all of your fixtures together in order to be able to do a PAN fan; TILT fanning can be done without panning.

To use the interpolate feature with the Fanning Tool

Note: Interpolation (or auto-arranging fixtures) only works on conventional fixtures in CAD mode.

In CAD Mode only, you have the option of using the Interpolate buttons too, which allow for a different type of Fanning Tool operation.

1 Select a group of fixtures by drag-selecting or control-clicking in either Wireframe or Shaded View (in DESIGN mode, this is part of the Focus Designer tool).

2 Take the first fixture and the last fixture in your selection and move them where you would like them as the first and last units in your fan.

3 When you click the Interpolate button for Pan and Tilt, the internal fixtures of your selection between the first and last will interpolate (or auto-arrange) themselves between those first and last units. Interpolate Pan will arrange the focuses of the internal fixtures' pan values; Interpolate Tilt will arrange the tilt values between the selection.

Fixture-specific focusing tools

All fixture-specific focusing tools such as shuttering, spotting/flooding, lens changes, and footprints are available through the selected fixture's properties on the *Shutter*, *Lens*, and *Fixture* pages respectively.

To make shutter cuts

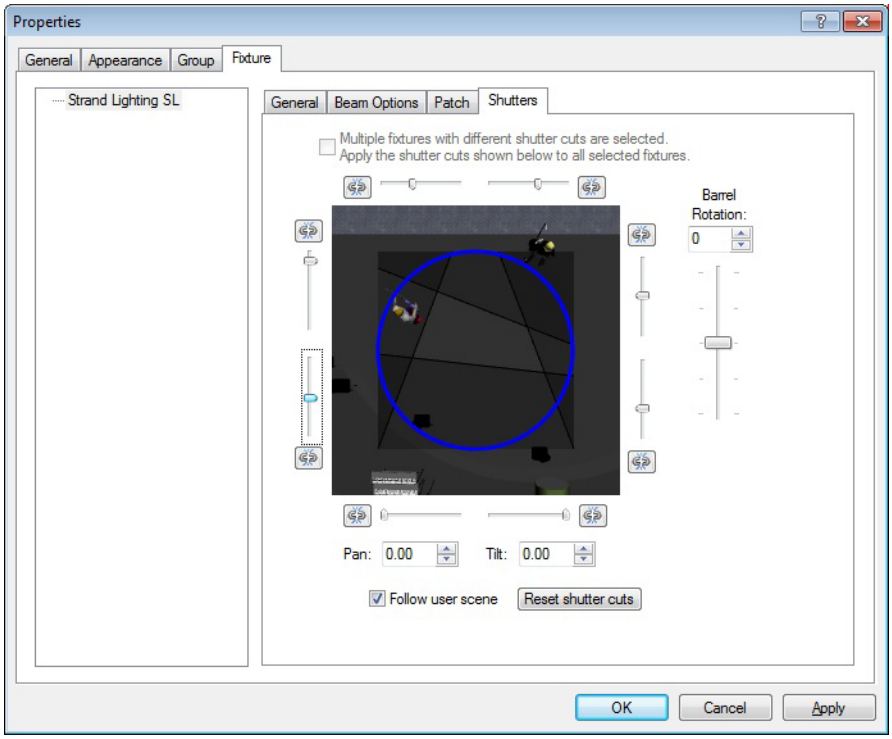
1 Select the fixtures for which you want to make shutter cuts.

2 Right-click and choose **Properties** or simply open the Shutter Cuts window.

3 Click the *Fixture* tab.

4 Click the *Shutters* tab.

Note: The POV image is not available when a fixture is zoomable or articulable.



5 Use the sliders along the top, bottom, and the sides to move the shutter blades.

Result: The cuts that you make are visible in the drawing.

Note: Click the small "lock" icon to lock a blade before or after moving one of its "ends." Once locked, the blades can be moved in or out from both "ends"/"sides" at the same time. If a blade is locked after moving only one of its "ends"/"sides", the blade will move in or out at an angle.

6 When applicable, use the *Barrel Rotation* slider to adjust the fixture's barrel orientation. You may also enter a barrel rotation angle or use the up/down arrow buttons to make these adjustments.

7 Adjust the pan and tilt of the selected fixture(s) by typing the appropriate values in the boxes in degrees, or by using the up and down arrow buttons.

8 Click *OK* when finished.

Note: If needed, click *Reset Shutter Cuts* to bring all the shutter cuts to the full-open position.

Tip: You can perform all of these steps with the Shutter Cuts tool, which is accessible from the Tools menu. The controls in this tool are identical to the ones described above in the Shutters tab; however, all changes made with the Shutter Cuts tool are applied instantly, without the need to click any other buttons.

To adjust the leaves of barndoors

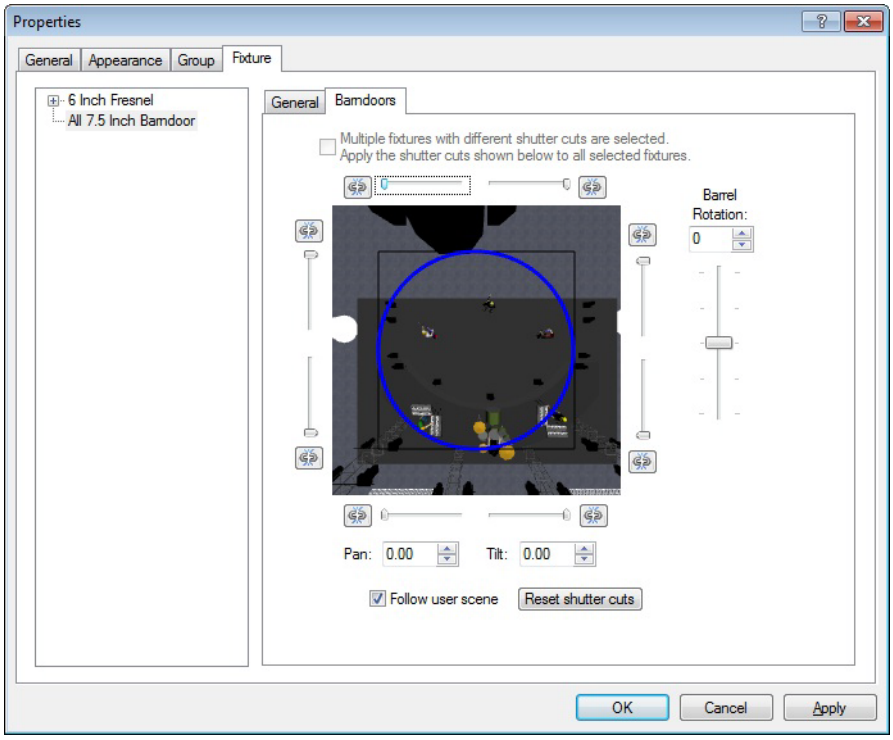
1 Select the fixture with barndoors for which you want to adjust the leaves.

2 Right-click and choose **Properties**.

3 Click the *Fixture* tab.

4 From the list on the left side, highlight the barndoor.

5 Click the *Barndoors* tab.



6 Use the sliders along the top, bottom, and sides to move the barndoor leaves. You will notice that the movement of the leaves is restricted to left, right, up, and down.

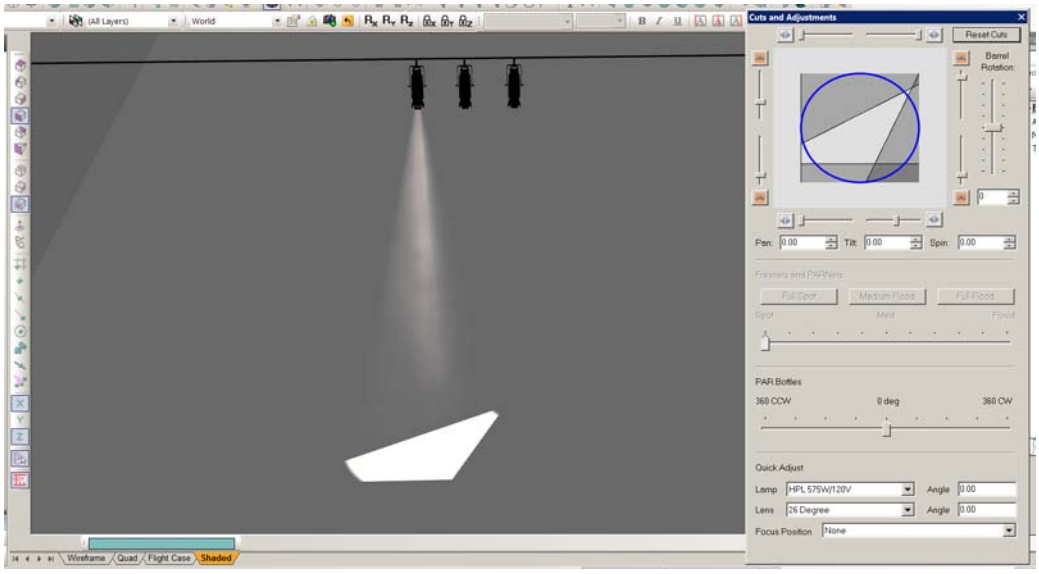
Result: The cuts are visible in the drawing.

7 Click *OK* when finished.

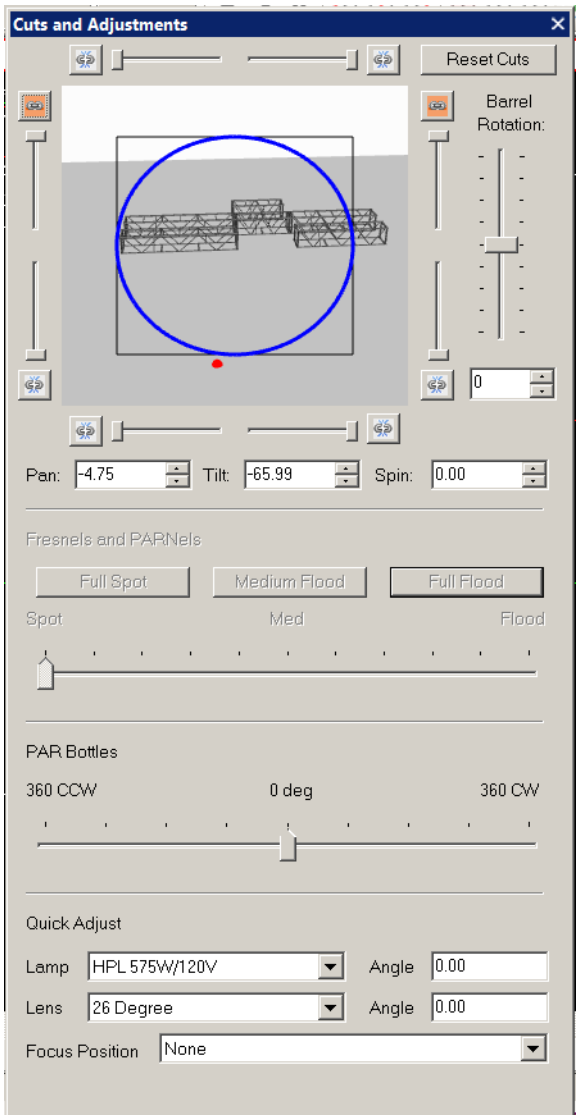
Using the Cuts and Adjustments tool

The Cuts and Adjustments Tool can handle a lot of Conventional changes dynamically, and gives you more controls for beam shaping. It is your one-stop tool for fixture beam shaping and edits.

Here's an example of the Cuts and Adjustments Tool in use:



Cuts and Adjustments tool properties

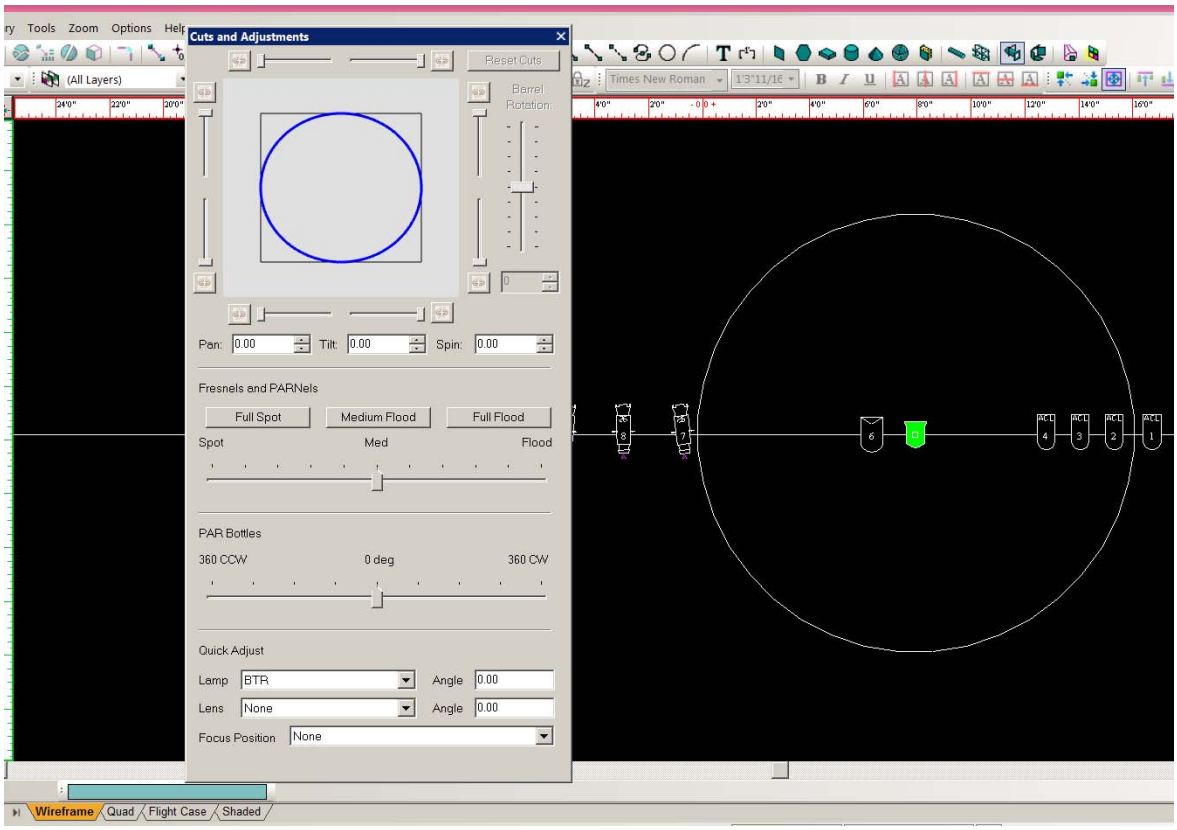


Shutters and Barrel Section

This section allows for shutter cuts on any fixture with shutters, as well as focusing the actual position of your fixture with accuracy. You can also quickly reset any and all of your shutter cuts, lock a blade on the Shutter mechanism, rotate your fixture's barrel, and spin the fixture's c-clamp on the lighting position.

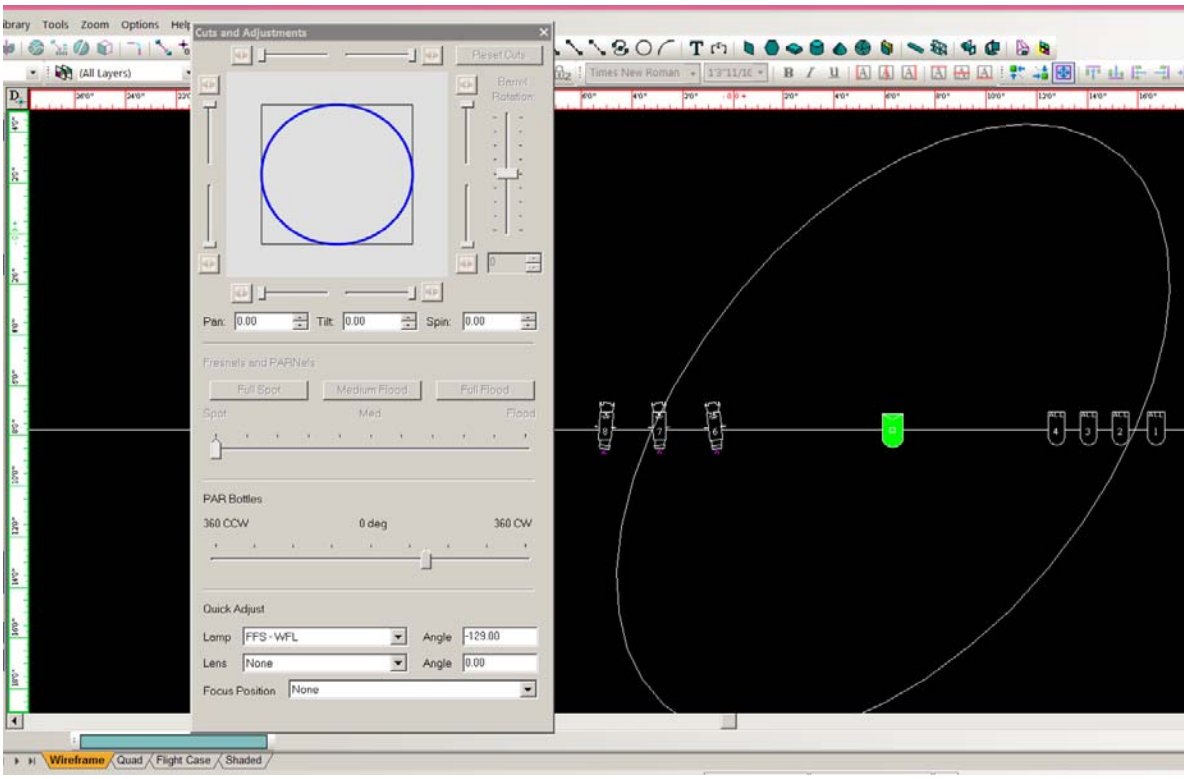
Fresnels and PARnels Section

This section of the Cuts and Adjustments Tool is for fixtures with a Fresnel-type lens or lamp train style body – the tool provides a slider for moving the lamp closer to the lens for wider angles and away from the lens for a tighter spot pool. You can also go Full Spot, Medium Flood, or Full Flood with the push of a button.



PAR Bottles Section

This section provides you with an easy access slider for rotating your PAR bottles. Quickly spin your PARs' lamp bottles to get the angle you need and the coverage you want to achieve.



Quick Adjust Section:

This section is copied from the Fixture Properties dialog so that you can change a lamp or a lens quickly and across multiple fixtures. You can also spin your lamps and lenses and reassign Focus Positions to any conventional fixture on your plot with the Cuts and Adjustments Tool's Quick Adjust section.

To open the Cuts and Adjustments tool

1 In Wireframe view, select the fixture that you want to adjust.

2 Click **Tools** > **Cuts and Adjustments**.

Result: The Cuts and Adjustments tool appears.

Manually adjust fixture attributes

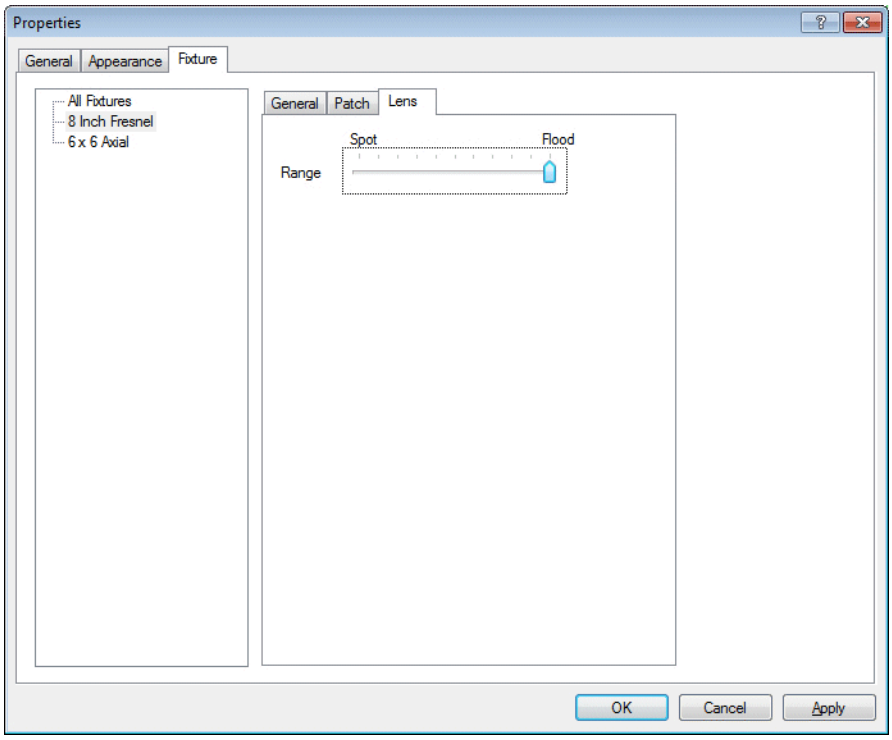
To adjust flood or spot

1 Select the fixture(s) for which you want to adjust the flood or spot.

2 Right-click and choose **Properties**.

3 Click the *Fixture* tab.

4 Click the *Lens* tab.



5 Use the slider to adjust the spot and flood.

6 Click *OK* when finished.

To change the bulb in a Par fixture

1 Select the PAR fixture that you want to focus.

2 Right-click and choose *Properties*.

3 Click the *Fixture* tab.

4 From the *Lamp* drop-down list, select the lamp that you want to focus.

5 In the *Angle* box for the lamp, type the rotation angle. The angle determines the degree of rotation of the lamp within the housing.

6 Click *OK* when finished.

Note: To focus Source 4 PAR fixtures, perform the same series of steps, except you must type a value in the *Lens Angle* box instead of the *Lamp Angle*.

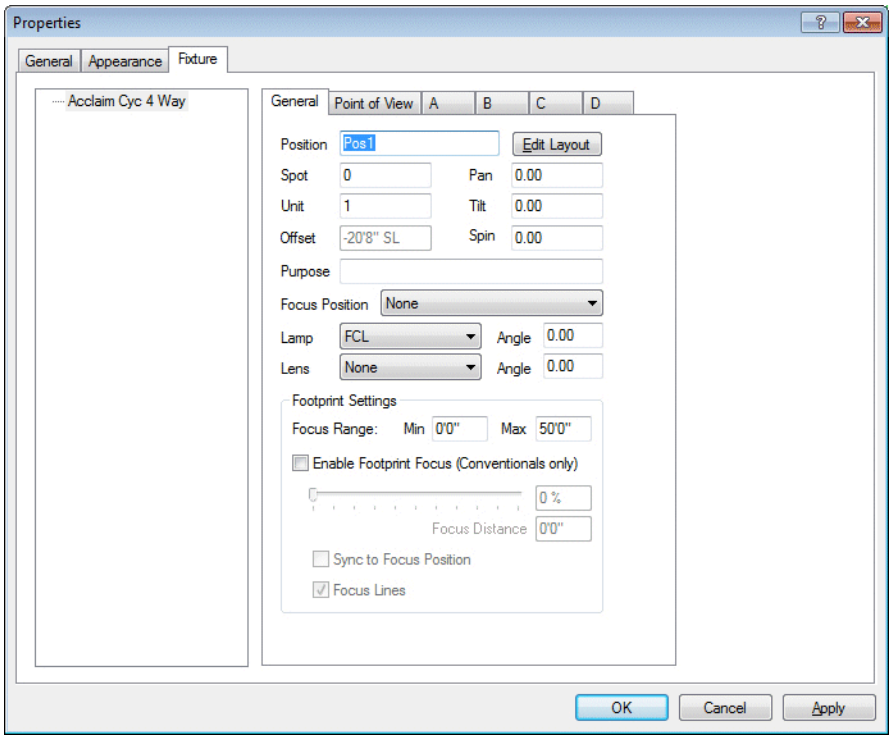
To focus a footprint for a conventional fixture

Note: This feature is also available for automated fixtures (in Design mode through the Footprint Designer Tool). For information, see page 481.

1 Select the conventional fixture.

2 Right-click and choose *Properties*.

3 Click the *Fixture* tab.



4 On the *General* tab, choose *Enable Footprint Focus (Conventionals only)*.

5 Adjust the fixture's minimum and maximum *Focus Range* in the *Min* and *Max* boxes.

6 Adjust the focus distance using the slider or by entering a value in the *Focus Distance* box.

7 Check *Sync to Focus Position* to adjust the footprint focus distance to the specified *Focus Position*.

8 Click *OK* when finished.

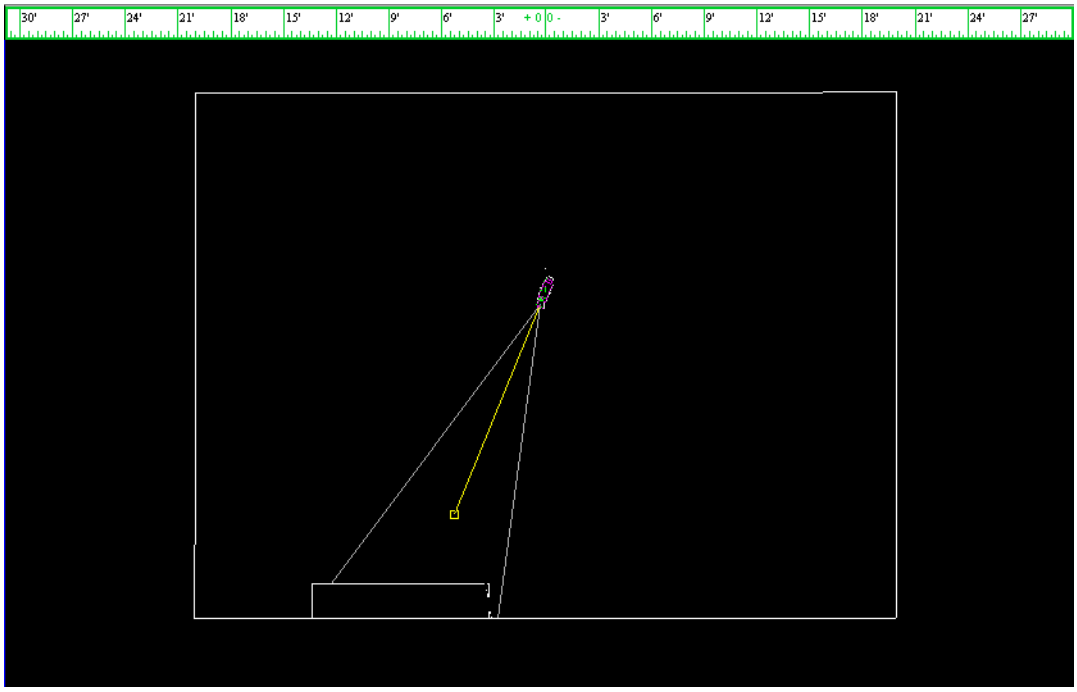
Result: The effect is visible in the Shaded view when *Footprint Focus* is enabled on the Lighting tab of the View Options window.

To focus a footprint for a conventional fixture (interactive method)

Note: This feature is also available for automated fixtures (in Design mode through the Footprint Designer Tool). For information, see page 481.

1 Select the conventional fixture.

2 Right-click and choose *Footprint Focus*. The fixture focus distance is represented by a yellow line in the CAD view.



3 Press the up and down arrows on your keyboard to adjust the focus distance.

4 When you are satisfied with the distance, right-click and select *Finish Footprint Focus*. To cancel your changes, right-click and select *Abort Footprint Focus*.

The effect is visible in the Shaded view when *Footprint Focus* is enabled on the Lighting tab of the View Options window.

Fixture properties

All fixtures in WYSIWYG have properties. Properties are the characteristics or attributes that shape and define a fixture.

Fixture properties can be divided into two categories:

- the properties that are common to all fixtures in WYSIWYG
- the properties that you define by assigning attributes to a fixture

Fixtures also have many non-visible attributes, some of which can be edited. These include Alias (found as Type in the reports), Cost, Weight, and AutoFocus codes.

General fixture properties

In WYSIWYG there are five tabs that appear in the properties dialog box for every fixture:

- Appearance
- Version
- Data
- Options
- Photometric

Each tab is explained in this section.

To view fixture properties that are common to all fixtures

1 From the **Library** menu, choose **Browse Library**.

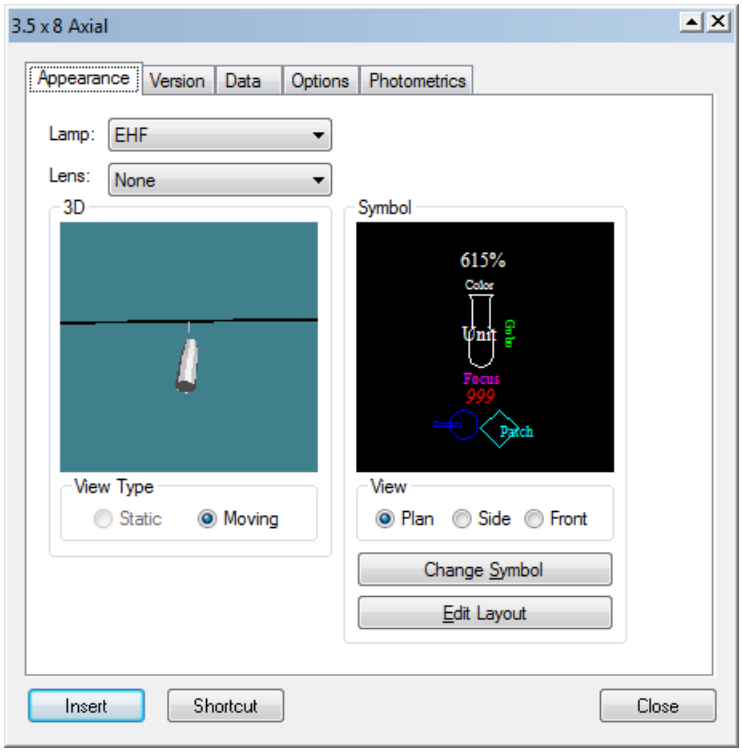
2 Click the *Fixture* tool at the bottom of the browser.

3 Browse to the desired fixture and click on it to select it.

4 Right-click on the fixture name, and then click **Properties**.

Tip: You can also click the *Properties* tool at the top of the Library Browser.

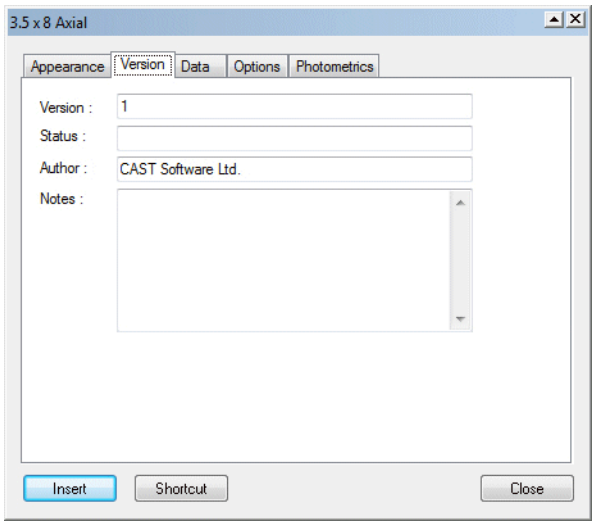
Result: The properties window for the selected fixture appears, with a shaded view and an image of the fixture's symbol displayed in the two boxes. An example is shown below:



5 Click *Plan*, *Side*, or *Front* to view the fixture in the different views. The Wireframe view changes based on the view type that you select.

Version tab

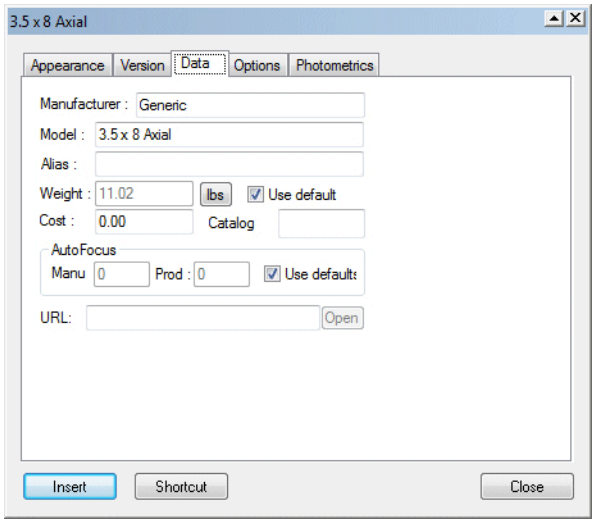
Options on the *Version* tab are read-only and are set when the fixture is added to the library.



- *Version*: The release number of the fixture.
- *Status*: The status of the fixture. For example, *Beta* indicates that the fixture is completed but not verified. *Release* indicates the fixture has been tested and is accurate. A status of *Preliminary* indicates that the fixture has minimal functionality.
- *Author*: The individual or company who constructed the fixture.
- *Notes*: Any notes that pertain to the fixture. The notes usually pertain to the simulation capabilities in WYSIWYG.

Data tab

Options on the *Data* tab are read-only and are set when the fixture is added to the library. This does not, however, include the *Alias*, *Weight*, *Cost*, *Manu*, and *Prod* fields. To modify these fields, access the desired fixture's properties from the Library Browser.



- *Manufacturer*: The name of the manufacturer of the fixture.
- *Model*: The model number of the fixture.
- *Alias*: The alternate name for the fixture.
- *Weight*: The weight of the fixture.
- *Use default*: Select this check box to use the default weight.
- *Cost:* The cost of renting or purchasing the fixture. The cost of renting the fixture can be the price for a day, week, or month. The default value is 0.00.

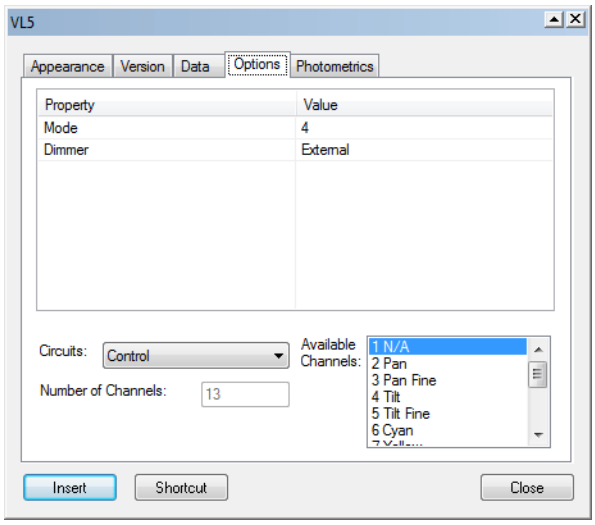
  Notes:

- If a cost is assigned to a fixture type when a WYSIWYG document is open, the cost is applied to the current document only.
- If a cost is applied to a fixture type when a document is not open, the cost is used as the default for all documents created on your copy of WYSIWYG.
- When opening a document created by someone other than yourself, the cost entered in that person's document is used.
- The cost entered in this tab is displayed in WYSIWYG reports. For more information on the display of this data, refer to page 359.

- *Catalog*: The code that is used by the manufacturer to identify the fixture.
- *Manu*: The manufacturer's code for those fixtures that are Autofocus-compatible. Conventional fixtures do not have a manufacturer's code. The default value is 0.
- *Prod*: The product code for those fixtures that are Autofocus-compatible. Conventional fixtures do not have a product code. The default value is 0.
- *Use default*: Select this check box to indicate that the default settings for Prod and Manu (0, 0) are to be used.
- *URL*: The internet address of the manufacturer.
- *Open*: If an internet address is shown, click this button to open the Web page of the manufacturer.

Options tab

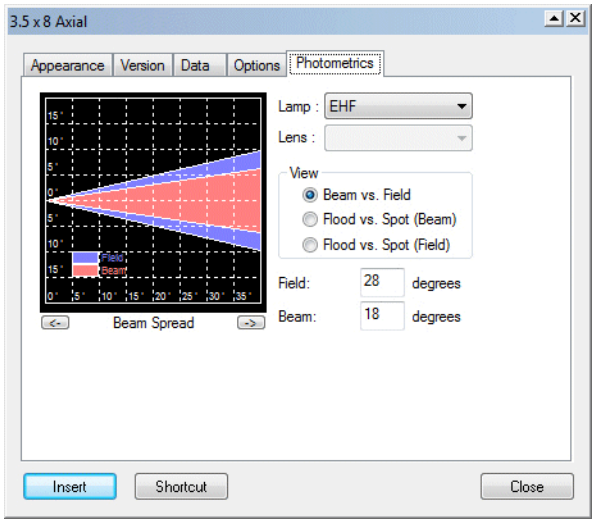
The boxes on this tab indicate various options about the selected fixture, such as the number of channels and the available channels. Based on the fixture selected, the options shown on this tab vary. For example, if you choose a light like the Vari*Lite VL5, the options on the left side of the window affect those on the right side. In this case, based on the options that you select on the left side, the hookup for this fixture may differ.



- *Mode*: Select the mode in which you want to configure the fixture. This field is not applicable for certain fixtures.
- *Dimmer*: If applicable, select the type of dimmer for this fixture (for example, choose either Internal or External).
- *Circuits*: Indicates the type of DMX controls for this fixture (for example, Control, Intensity, RIntensity, Color, and so on).
- *Number of Channels*: Based on the circuit that you selected above (in addition to the options selected), this box indicates the number of channels for this fixture.
- *Available Channels*: The names of the channels available for this fixture (based on the circuit and options that you selected).

Photometrics tab

Options on the *Photometrics* tab affect the lamp and lens settings for the fixture.



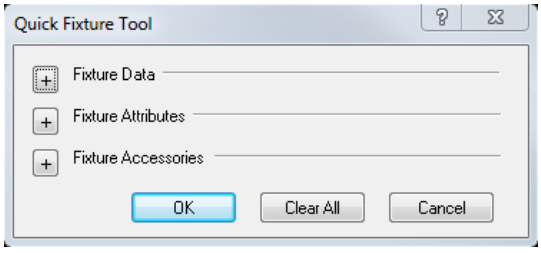
- *Lamp*: Select the lamp to be used in the fixture.
- *Lens*: If applicable, select the lens setting for the fixture. Changing the lens alters the photometric properties of the fixture.
- *View*: Select the graphical representation that you want to display in the box to the left. The choices are as follows:
  - *Beam vs. Field*. Displays a graphical representation of the beam and field angles of the fixture.
  - *Flood vs. Spot (Beam)*. Displays a graphical representation of the ratio between the flood and spot for the fixture's beam angle.
  - *Flood vs. Spot (Field)*. Displays a graphical representation of the ratio between the flood and spot for the fixture's field angle.
- *Field*: The measurement in degrees from the center of the beam to where the light level drops off to 10 percent of the peak.
- *Beam*: The measurement in degrees from the center of the beam to where the light level drops off to 50 percent of the peak.

Quick Fixture tools

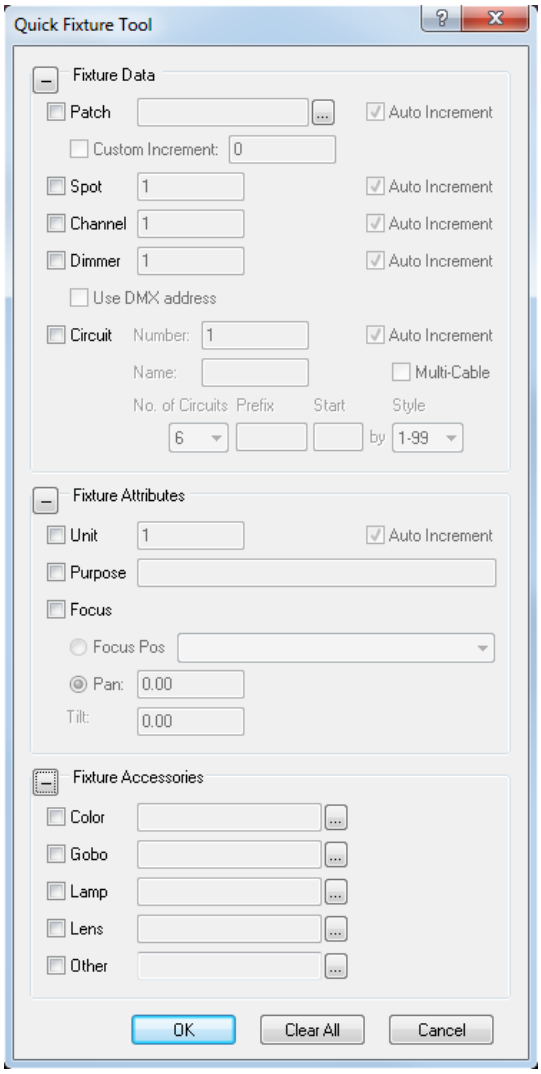
Quick Tools allows you to quickly assign data, accessories and other important attributes to your lighting fixtures without having to switch back and forth between DATA mode or opening each fixture's properties for each change. Quick Tools will enter in repetitive information, incrementing automatically if you choose, for all of your fixtures.

The Quick Tools window allows you to expand or collapse each section of the Quick Fixture Tool as you see fit to improve your workflow and space.

Here is the Quick Fixture Tool completely collapsed:



Here is the Quick Fixture Tool completely expanded:



Quick Fixture Tool Sections

When you enter data into the field associated with each value and click *OK*, a wireframe view appears, where you will click on the fixtures to which you would like to assign the changes you made in the tool.

Fixture Data Section

This section allows you to assign repetitive values for Spot, Channel and Dimmer (and Auto Increment), DMX addresses per Universe with Auto Increment, and a Custom Increment Patch that allows you to assign addresses based solely on your custom increment and ignoring the logical patch increments.

The Fixture Data section also allows you to assign repetitive Circuit Names and Numbers to your fixtures for any situation – an example of this would be labeling Socapex circuits on a touring lighting truss A-1 through A-6. If the Multi-Cable option is enabled, Quick Tools will automatically increment the Circuit Name and reset the Circuit Number after every X fixture is placed, where X is the No. of Circuits selected.

Fixture Attributes Section

This is where you can assign repetitive values for Unit numbers (with Auto Increment if desired) as well as Purpose and Focus (either by Position or by degrees).

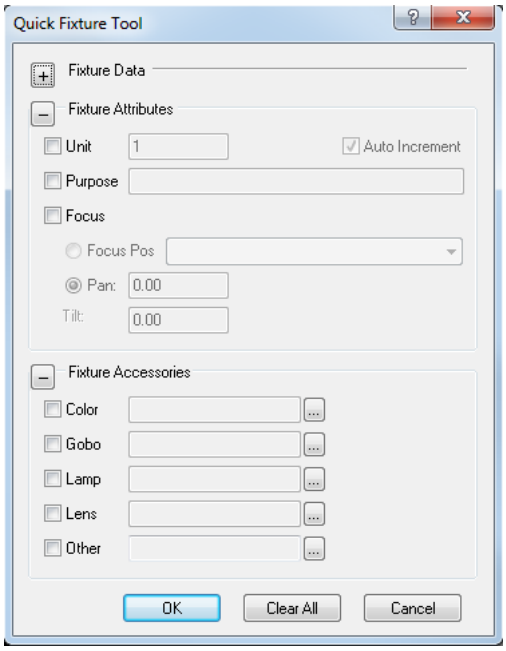
Fixture Accessories Section

This section of the Quick Fixture Tools dialog allows you to assign repetitive values for Color, Gobos, Lamps, Lens sets, and Other accessories as assigned.

Using the Quick Fixture Tools panel

1 From the **Tools** menu, choose **Quick Fixture Tools.** When the Quick Fixture Tools dialog opens, it defaults to the collapsed view. Click on the plus symbol next to each section to open it.

Tip: You can also click the Quick Tools tool on the Tools toolbar.



2 To activate a tool, select the check box to the left of the tool. You can activate as many as you want at a time. See below for explanations of the different tools.

3 Type the initial value for the tool in the box to the right of the tool name. If you want the value to auto increment, as for channels or spot numbers, make sure you select the *Auto Increment* check box. WYSIWYG will assign the next sequential number based on the properties and requirements of the previous fixture.

4 When the dialog box is set up the way you want it, click *OK*.

Result: The cursor will show a “Q”, indicating that Quick Tools have been activated.

5 Click on each fixture in the order you want any auto-incrementing values to be assigned.

Note: The attributes to be assigned to the next fixture are logged in the bottom left hand corner of the status bar.

6 When complete, right-click and choose *Finish Quick Tools*.

Note: To change the values at any time, right-click and choose *Change Options*.

Tip: Use the *Clear All* button to reset all the values and deselect all quick tools.

Notes:

- When typing values, type the exact value or use the [...] browse button to open a selection dialog.
- The entry in the patch box must fulfill the requirements of patch notation which is universe.#. For more information regarding patch notation refer to “Reading the patch” on page 372.
- For more information about the data fields, refer to “Data fields” on page 358.
- When using the quick tools to insert a color or gobo, WYSIWYG automatically inserts a default color frame or gobo holder, which is already defined in the fixture. Remove the default frame and add a new one using Add Accessory. For information on adding accessories, refer to “To insert an accessory” on page 315.
- When you are applying a color to a multi-source fixture, a dialog appears, listing the circuit names to which you can apply the Quick Tool color selection. To apply the same color to multiple circuits listed in this dialog, press CTRL, and then select the desired circuits. Press *Select* when you are finished.

Fixture-specific properties

Customize the properties of a fixture by assigning specific attributes, such as point of view, patch information, and lens type.

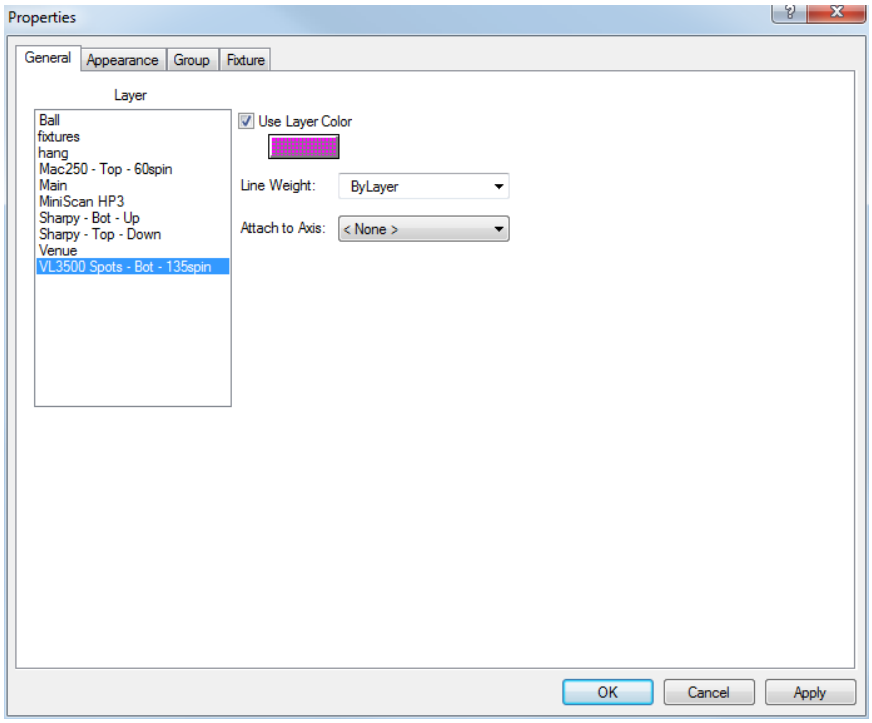
To edit / view fixture attributes

1 Select the fixture that you want to view or edit.

2 Right-click and select **Properties**.

Tip: At any time, to access an object’s properties, you can click on the *Properties* tool on the *Edit* toolbar. 

Result: The *Properties* dialog box is displayed. Based on the type of fixture that you have selected, the Properties dialog box varies. An example is shown in the graphic below.



Fixture tab

Options on the *Fixture* tab affect the fixture properties and their accessories. On the left side of the dialog box is a list of the selected fixtures and their accessories. Click on a list item to obtain further details about the item.

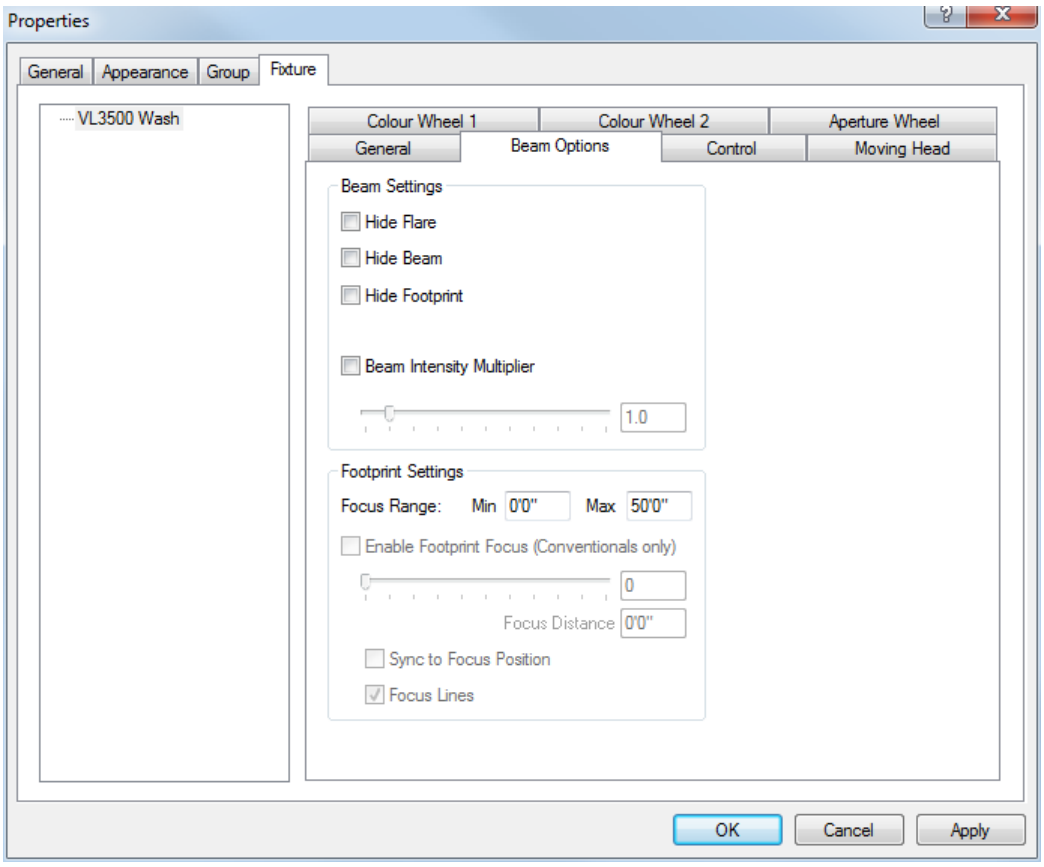
General tab

Refer to “Data fields” on page 358 for a description of the fields.

- *Edit Layout*: Click to edit the text label layout for the selected fixture. For more information on layout refer to “Fixture label layout” on page 310.

Beam Options tab

Options on the Beam Options tab affect the beam and footprint settings for the selected fixture.



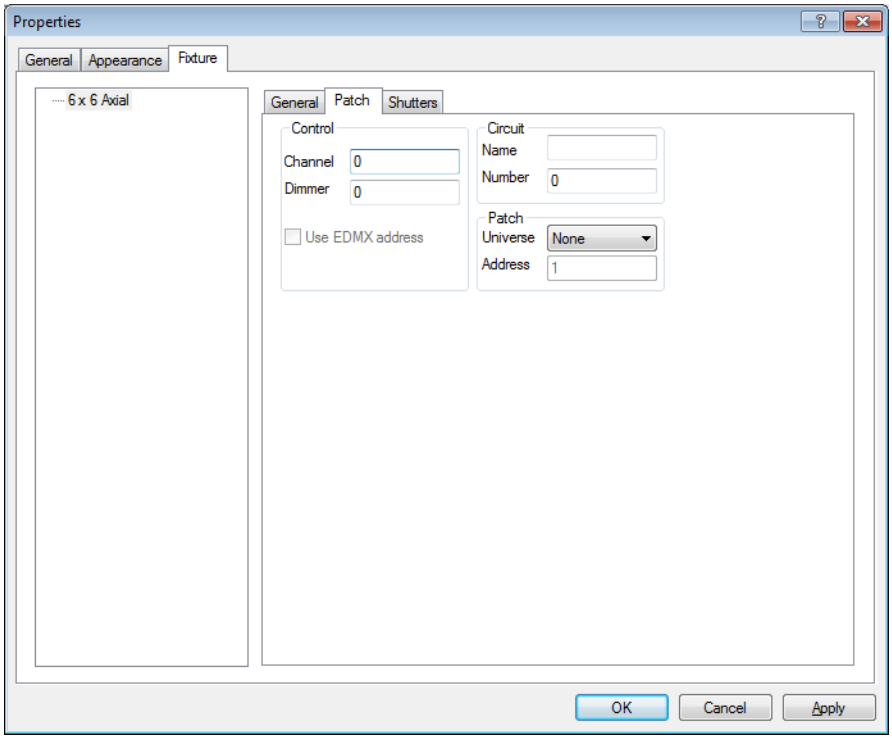
- *Beam Settings*: Use these options to control whether you want the beam, footprint, or flare to be visible when turned on in Shaded View. The settings here will override the Shaded View > View Options settings.
  - *Hide Flare*: Disables the flare for this fixture in Shaded View. The Source disk will still be visible.
  - *Hide Beam*: Disables the beam for this fixture in Shaded View.
  - *Hide Footprint*: Disables the footprint for this fixture in Shaded View.
  - *Beam Intensity Multiplier*: By default this value is set to 1.0 to use the default values of the Fixture's beam, as defined by the library. To increase or decrease the beam's brightness, please adjust using the slider. For example, to double the brightness of the beam, set the slider to 2.0.

  Note: Any changes will be applied to all fixtures of this type in the file.
- *Footprint Settings*: Use these options to control the footprint focus on conventional fixtures.

- *Focus Range*: Set the minimum (*Min*) and maximum (*Max*) focus distances for the fixture.
- *Enable Footprint Focus (Conventionals only)*: Check this option to control the footprint focus.
- *Focus Distance*: Set the focus distance for the fixture.
- *Sync to Focus Position*: Check this option to adjust the footprint focus distance to the specified *Focus Position*.
- *Focus Lines*: Check this option to display focus lines for the selected focus in Shaded Views.

Patch tab

Options on the *Patch* sub-tab affect the circuit, channels, and patch universe for the selected fixture.



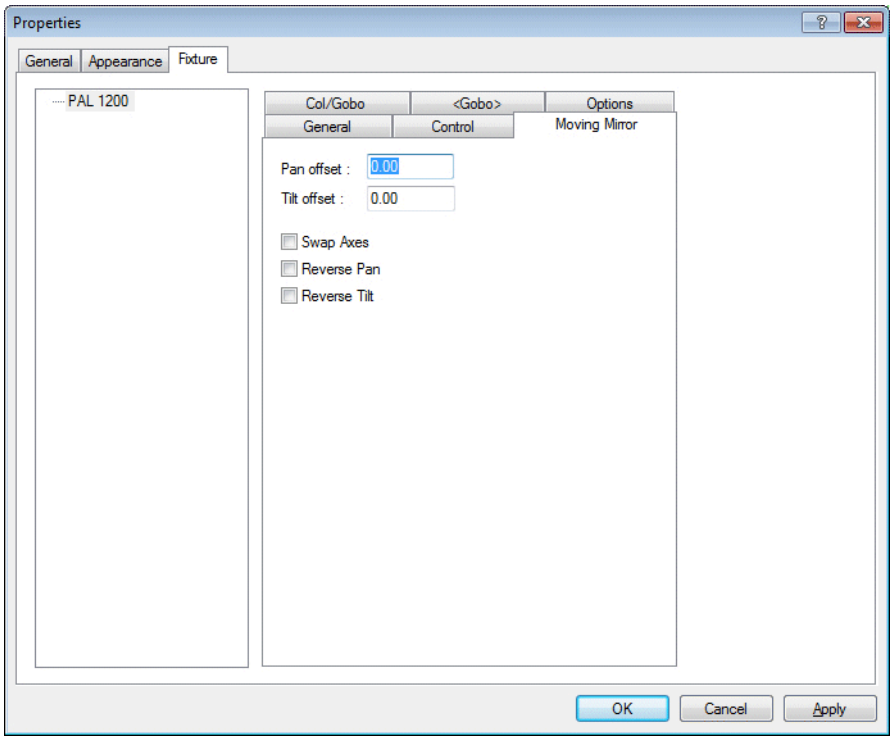
- *Control, Circuit, and Patch*: Refer to “Data fields” on page 358 for a description of these boxes.
- *Use EDMX address:* Select this check box to use the EDMX address for the patch universe. This check box is enabled if you select EDMX from the *Universe* drop-down list. EDMX is an Ethernet protocol developed by ETC.

Shutters tab

For information on the Shutters tab, see “To make shutter cuts” on page 288.

Moving mirror / moving head tab

Options on the *Moving Mirror* and *Moving Head* sub-tabs affect the pan and tilt values.

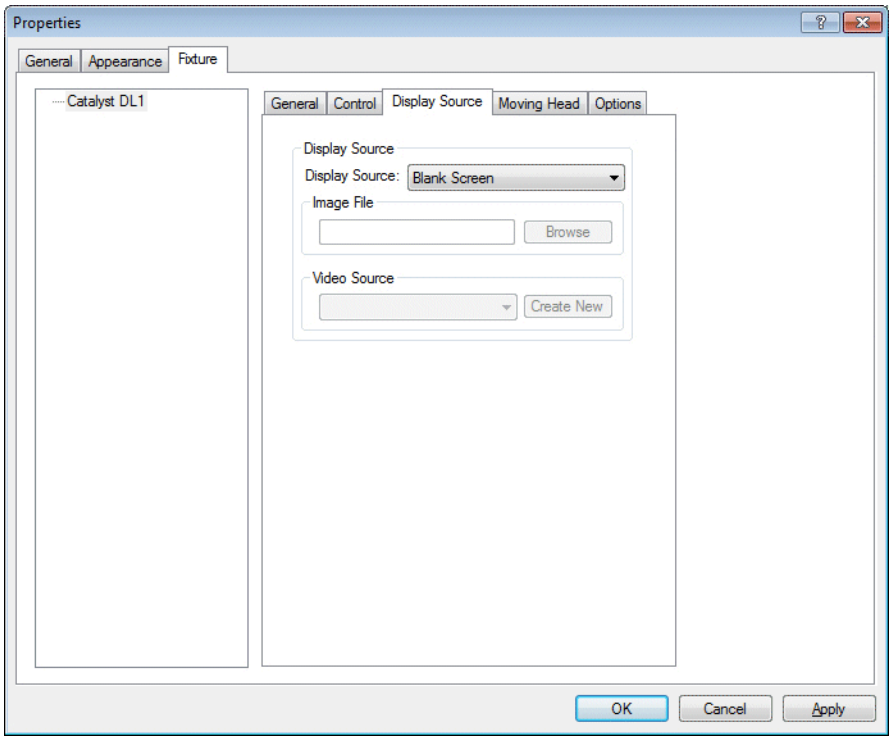


- *Pan offset*: Sets the manually adjustable pan angle.
- *Tilt offset*: Sets the manually adjustable tilt angle.
- *Reverse Pan:* Select this check box only if the real fixtures have an option to reverse the pan parameter and that option is set to true.
- *Reverse Tilt:* Select this check box only if the real fixtures have an option to reverse the tilt parameter and that option is set to true.
- *Swap Axes:* Select this check box only if the real fixtures have an option to swap the control channels for the tilt and pan and that option is set to true.
- *Pan*: (Only visible for certain fixtures.) Displays the manufacturer low, mid and high pan limit values for the fixture. To change a value, select it, click *Change*, and then type the new value.
- *Tilt*: (Only visible for certain fixtures.) Displays the manufacturer's low, mid, and high tilt limit values for the fixture. To change a value, select it, click *Change*, and then type the new value.

  Note: The Pan and Tilt limit behavior is defined by the Mode selected on the Options tab.

Display Source tab

Options on the *Display Source* sub-tab are used to add a video source to a Video Projection fixture, so that it can display a video source in the fixture's beam.



Select the *Display Source*.

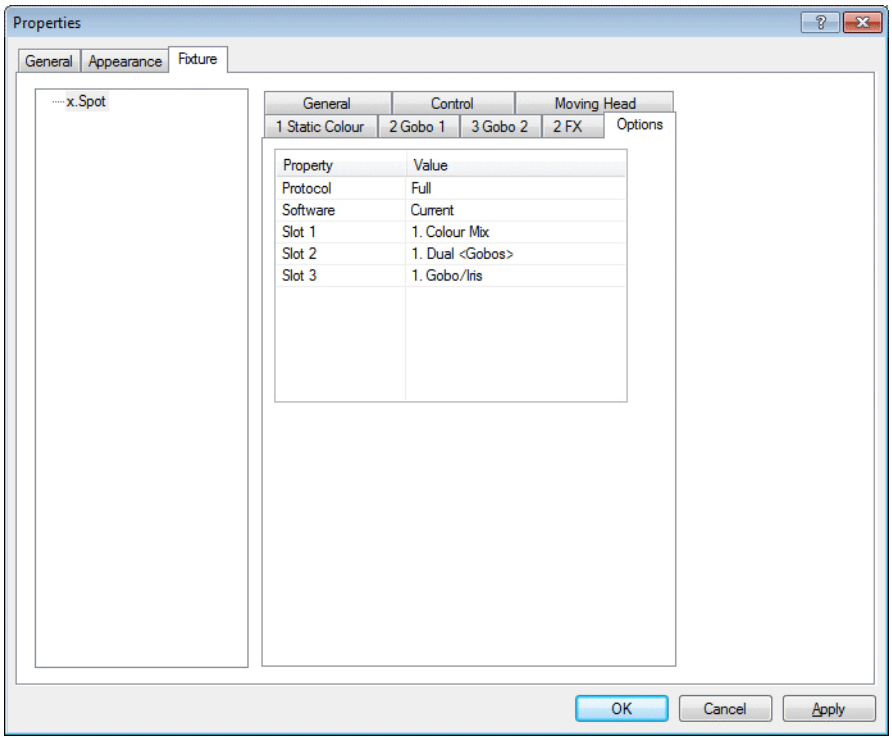
- To leave the screen blank, select *Blank Screen*.
- To show a static image on the screen, select *Image File*, and then click *Browse* to locate the graphic. (The image must be saved in either bitmap or JPEG format.)
- To attach a video source to the screen, select *Video Source*, and then choose the video source from the drop-down box (or click *Create New* to configure a new video source with the Video Manager). For details on creating a new video source, see “To configure a new video source for streaming video” on page 341.

Notes:

- For details on patching the video control in Data mode, see “To patch the control of a video source” on page 369.
- For details on using the Video Designer Tool to play the video in Design mode, see “To use the Video tool” on page 480.
- For details on using a console device to control the video in Live mode, see “To control a DMX patched video source with a console” on page 531.

Options tab

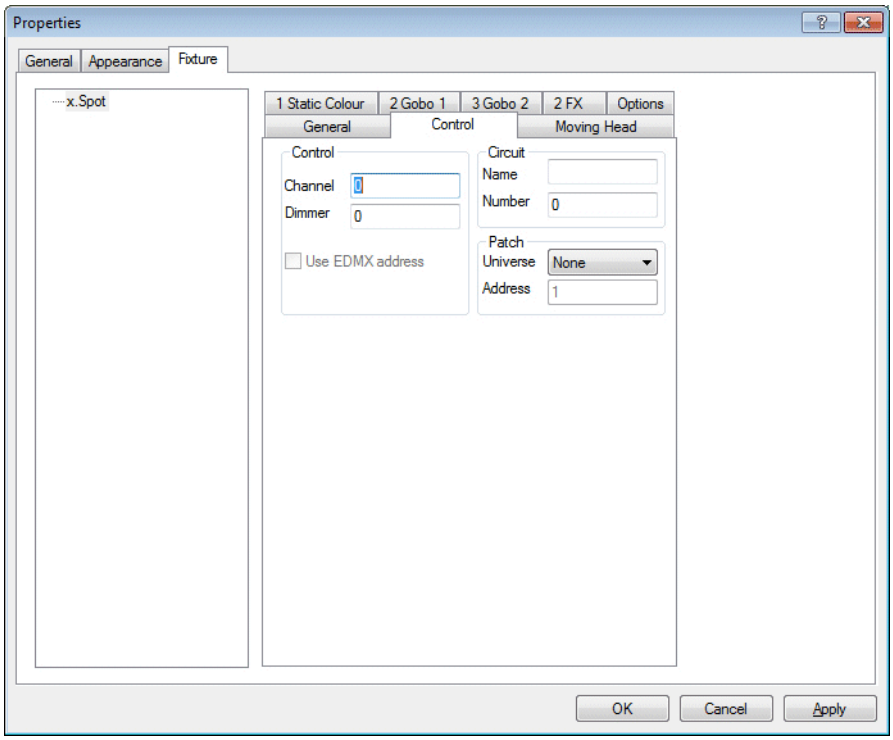
Options on this tab affect the mode selection for the fixtures. The options vary according to the fixture that you have selected.



- *Mode*: Sets the mode for the selected fixture.
- *Software:* The software version that is loaded on the selected fixture.
- *Slots*: The mechanical components that are present in the selected fixture.
- *Control:* Simulates the lamp control channel on the selected fixture. Since you can physically turn a fixture on or off, WYSIWYG enables this action in the software.

Control tab

Options on the *Control* tab enable you to set the channel, circuit, and patch universe information for the fixture.



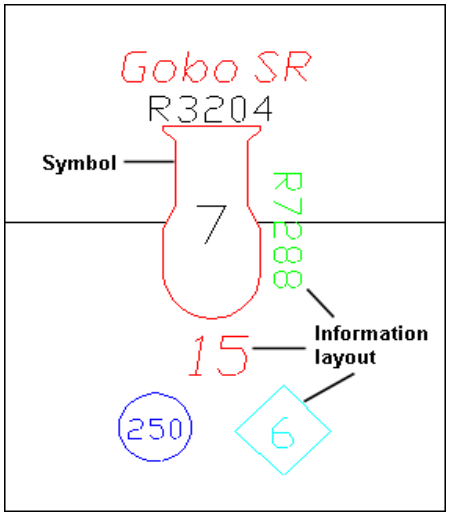
- *Channel*: Type the channel number for the selected fixture.

- *Dimmer*: The dimmer number for the selected fixture.
- *Use EDMX address*: If checked, the EDMX address is used for the patch universe. This check box is enabled if either EDMX or Conventionals (for the DMX address) is selected from the *Universe* drop-down list.
- *Name*: The circuit name for the selected fixture.
- *Number*: The circuit number for the selected fixture.
- *Universe*: Select a patch universe for the fixture.
- *Address*: The starting DMX channel number for the selected fixture.

Fixture notation

Fixture notation affects how fixtures and their properties are displayed on your drawing and consequently on your plot in Presentation mode. As illustrated below, two components comprise fixture notation:

- symbol
- fixture label layout



Symbols

A symbol is a graphical representation of a fixture in a Wireframe view. In all wireframe views except isometric, symbols may be used to represent fixtures instead of three dimensional models. Fixtures are associated with specific symbols by default, but you can change the symbol to whatever you like.

To turn fixture symbols on or off

1 From the **Options** menu, choose **User Options**.

Result: The *User Options* dialog box is displayed.

2 Select the *Show Details* tab.

3 Select the *Draw Symbols* check box.

4 Click *OK*.

To change the symbol for a fixture type

The following procedure changes the symbol for all instances of the selected fixture type in the document. Similar procedures change the symbol at an application level.

Notes:

- To change fixtures at an application level, see “To change the symbol for a fixture at the application level” below.
- For fixtures with interchangeable lenses (or lamps in the case of PARs), you must change the symbol for the lamp or lens instead of the fixture.

1 From the **Library** menu, choose **Browse Library**.

2 Select the Fixtures library.

3 Navigate through the library, and then select the fixture type for which you wish to change the symbol.

4 Click the *Property* tool.

Result: The properties dialog for fixture type opens.

- When modifying the symbol for a specific fixture and lamp combination, use the Lamp drop-down to select a specific Lamp for the fixture.
- When modifying the symbol for a specific fixture and lens combination, use the Lens drop-down to select a specific Lens for the fixture.

5 Click *Change Symbol*.

Result: The symbol selection window is displayed.

6 Select the new symbol.

7 Click *Select*.

8 Close the fixture property box.

To change the symbol for a fixture at the application level

Perform this procedure if you want to modify fixture symbols universally so that when you create new files, the symbols that you want are displayed for fixtures (instead of the default symbols).

1 Start WYSIWYG and from the welcome window, select **Options** > **Browse Library**.

2 Click the Fixtures tab.

3 Navigate through the library, and then select the fixture type for which you wish to change the symbol.

4 Right-click the Fixture name, and then select **Properties**.

Result: The properties dialog for fixture type opens.

5 Click the view that you want to change.

6 Click *Change Symbol*.

Note: When the same symbol is used for two view types, the *Change Symbol* button is disabled for one of the view types. Click either of the other two view types to choose a new symbol.

Result: The symbol selection window is displayed.

7 Select the new symbol, and then click *Select*.

8 Close the fixture property box.

Note: Your changes are stored in the file *Default.ldb*, which is stored in the *WYSIWYG\Library* folder (in the location where you installed WYSIWYG). If you need to upgrade or reinstall WYSIWYG and you want to keep the changes you have made to symbols, first make a backup of Default.ldb file. Then, once the upgrade or reinstallation is complete, paste the backed up copy of the Default.ldb file into the *WYSIWYG\Library* folder, replacing the existing file.

Fixture label layout

You can place the following attribute labels around fixture symbols in WYSIWYG:

- focus
- lamp
- lens
- notes
- position
- purpose
- spot
- unit
- channel
- circuit
- color
- gobo
- patch
- dimmer

To set the position, visibility, color, and justification of these attributes, edit the Label Layout.

Note: To see which options you have turned on, you may need to enable the master visibility toggle from View Options > Show Details tab before the specified label layout appears in the wireframe. For details, see the “Show Details tab” on page 448.

As with symbols, you can edit label layouts at an application level, document level, or selected fixture-only level. To change the font used for the attributes, see “To change the font in Label Layouts” on page 313.

To change the label layout at the application level

To change the label layout at an application level, you must browse the library from the WYSIWYG Welcome window. Note that changes you make to a fixture’s label layout at the application level will appear for that fixture whenever a new WYSIWYG file is created.

Note: Since the information is stored in the Default.ldb file in the C:\ProgramData\CAST Software\WYSIWYG\1.xx.xx.x\Library folder, it is recommended that you make a backup of this file so you can keep your changes if ever you need to reinstall WYSIWYG. In this case, simply replace the installed file with the backup to ensure your changes are not lost. Alternatively, you can use the Export User Data feature to back up your settings. For details, see “Exporting User Data” on page 53.

1. In the Welcome window, select **Options** > **Browse Library**.
2. Select the Fixtures tab.
3. Navigate through the fixtures library, and then select the fixture type for which you wish to change the label layout.
4. Right-click and select **Properties**.

   Result: The Properties dialog box for the fixture is displayed.
5. Click *Edit Layout*.

   Result: The Label Layout window appears.
6. Edit the label layout as desired, and then click *OK* to save your changes.

To change the label layout at the document Level

To change the label layout at the document level, you must browse the library from within a WYSIWYG file. Any changes made to the fixture's label layout at the document level will only be applied to fixtures being inserted in the current file. New files containing the same fixtures will not be affected.

1 In your WYSIWYG file, select **Library** > **Browse Library**.

2 Select the Fixtures tab.

3 Navigate through the fixtures library, and then select the fixture type for which you wish to change the label layout.

4 Right-click and select **Properties**.

Result: The Properties dialog box for the fixture is displayed.

5 Click *Edit Layout*.

Result: The Label Layout window appears.

6 Edit the label layout as desired, and then click *OK* to save your changes.

To change the label layout at the per-fixture level

Changes that you make to a fixture's label layout at the per-fixture level only affect the selected instances of the fixture in the current file. There are two ways to change the label layout in this manner:

To modify any fixture's layout in the current file

1 In a wireframe view, right-click on an empty area and select **Fixture Attribute Layouts**.

Result: The Label Layout window appears.

2 Edit the label layout as desired, and then click *OK* to save your changes.

To modify a specific set of fixture label layouts in the current file

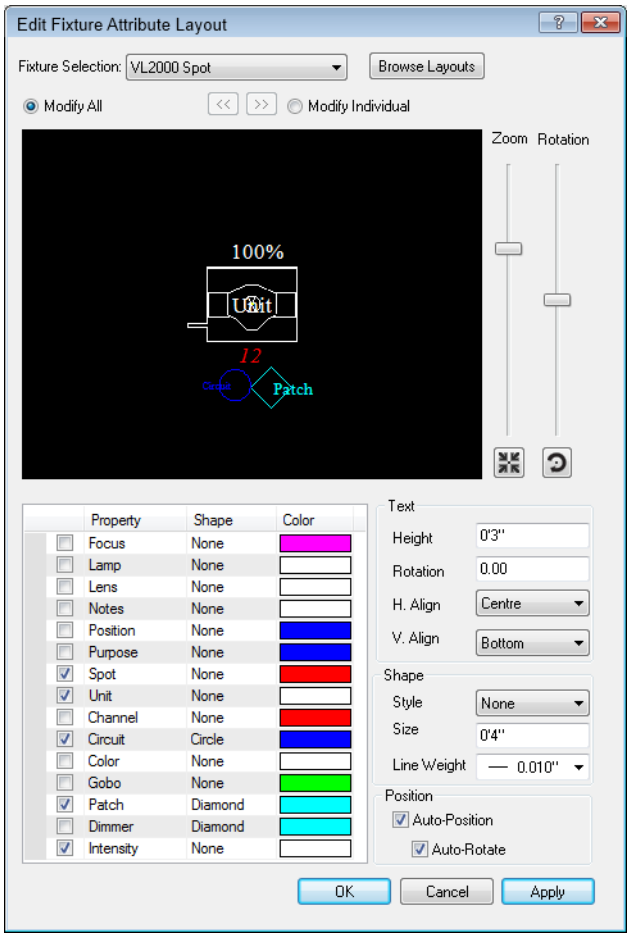
1 In a wireframe view, select the group of fixtures that you want to modify.

2 Right-click and select **Fixture Attribute Layouts**.

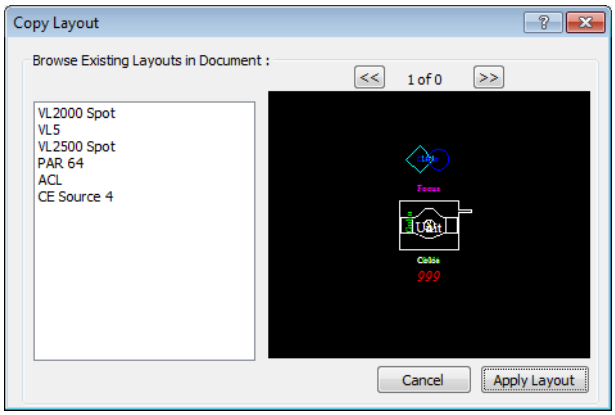
Result: The Label Layout window appears.

3 Edit the label layout as desired, and then click *OK* to save your changes.

Using the Edit Label Layout window



- *Fixture Selection*: Use this drop-down to select the fixture that you want to modify. This function is only available when accessing the Edit Label Layout window from the wireframe. With no fixtures selected, the Fixture Selection drop-down will list all the fixture types in the current WYSIWYG file. With fixtures selected, the Fixture Selection drop-down will list all fixture types in the selection set.
- *Browse Layouts*: Click this button to copy a layout from another fixture in the file. When you click this button, the Copy Template window appears:



From the list on the left, select the fixture layout you would like to copy, and then select the specific fixture using the arrow selectors above the preview on the right. When you have selected the fixture layout that you want to copy, click *Apply Layout*.

- *Modify Selector*:

- With *Modify All* selected, the changes that you make to the fixture's label layout are applied to all of the fixtures of the specified type within the selection set. If no fixtures are selected, the changes will be applied to all fixtures of that type in the file. To show you which fixtures will be affected, the wireframe view highlights and selects all affected fixtures.
- With *Modify Individual* selected, you can modify a specific fixture in the file. Use the arrows to cycle through the available fixtures, noting the preview image in the wireframe as it highlights and selects the corresponding fixture that you will be modifying.

- *Fixture Preview*: A wireframe preview of the fixture's label layout.
- *Zoom*: Adjust the *Zoom* slider to zoom the fixture's preview in and out. Click the Zoom Reset button, identified by this icon  to reset the zoom back to its original levels.
- *Rotation*: Adjust the *Rotation* slider to rotate the fixture and preview the behavior of the labels at various rotation angles. Click the Rotate Reset button, identified by this icon  to reset the fixture rotation back to its original levels.
- Adjust Labels
  - Use the checkboxes to turn various properties on/off. Note that properties will not appear even if you turn them on in this window, unless you enable the master visibility toggle in the Show Details window. For more information, see the "Show Details tab" on page 448.
  - The *Shape* column indicates the current shape assigned to the label.
  - The *Color* column indicates the current color assigned to the label. You can change the color by selecting the Color preview and using the drop-down to select a new color.
- Label Text
  - *Height*: Increase or decrease the height to modify the font size.
  - *Rotation*: Set the angle to specify the label alignment in relationship to the fixture symbol.
  - *H Align*: Set the horizontal alignment relative to the insertion point/shape.
  - *V Align*: Set the vertical alignment relative to the insertion point/shape.
- Label Shape
  - *Shape*: Select a shape to be associated with this attribute, if desired. You can choose from none, circle, square, octagon, or diamond.
  - *Size*: Increase or decrease the size of the shape, as desired.
- Label Position
  - *Auto-Position*: Select this option if you want the attribute's label to maintain its position in relation to the fixture symbol regardless of where the fixture is focused.
  - *Auto-Rotate*: Select this option if you want the attribute's label to maintain its alignment in relation to the fixture symbol regardless of where the fixture is used.

To change the font in Label Layouts

You can change the font that will appear globally in all Label Layouts, choosing from all fonts installed on your computer.

1. In any drawing mode, click **Options** > **User Options**.
2. Click the Font Styles tab.
3. Under Fixture Layout Labels, from the *Font* drop-down list, select the font that you want to appear in all Fixture Layout labels.
4. Choose whether you want the letters to appear bold or underlined.
5. Click *OK*.

Inserting symbols

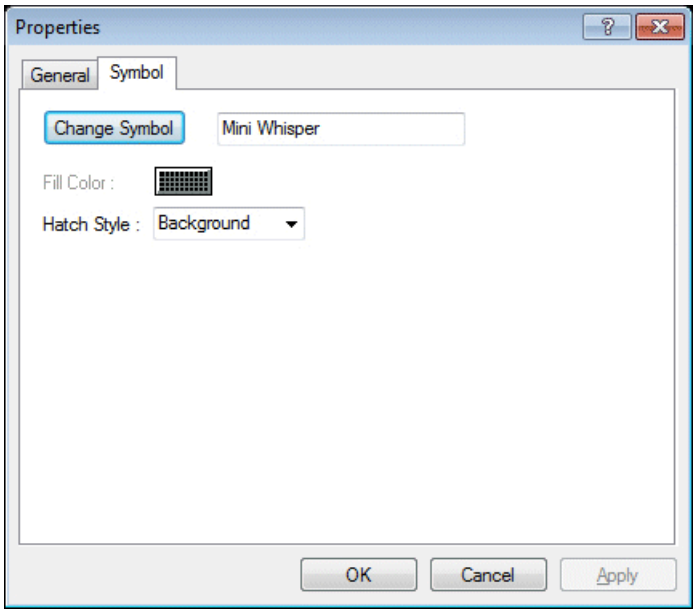
You can also insert symbols that are not associated with fixtures. This allows you to represent the fixtures when you draw a key or want to show the position of a boom in a plan view. Symbols are not fixtures and, therefore, are not counted in reports. Standard symbols, such as the United States Institute for Theatre Technology (USITT) symbols, are available from the library for your use.

To insert symbols

Refer to “The Library Browser” on page 133. For information on inserting objects from the library, see “To insert objects from the library” on page 135.

Symbol properties

Options on the *Symbol* tab affect the fill color and hatch style of a symbol.



- *Change Symbol*: Click to select a different symbol.
- *Hatch Style*: The style to use for the hatch. If the Hatch style is *None* the symbol is transparent. If it is *Background*, the symbol is filled with the color of the background and is not transparent.
- *Fill Color*: The color with which you want the symbol to be filled. If the hatch style is *None* or *Background*, the fill color is not enabled.

Color, gobos, and accessories

Introduction

You can add to fixtures accessories such as barndoors, tophats, color scrollers, dousers, and so on. Accessories have their own section in the Library Browser and can be inserted just like other objects. You must insert accessories onto existing fixtures and can only insert them onto compatible fixtures.

Mountable yokes

Note the following before working with mountable yokes:

- While mountable yokes are considered accessories, they differ from other accessories in that you need to add them before you add the fixtures (you cannot insert a fixture, and then select *Add Accessory* to add a mountable yoke).
- You cannot use the *Add Accessory* option to add a fixture on a recently inserted mountable yoke. Instead, you must add the fixture either by using a shortcut or by selecting it from the library browser. Then you can use the *Add Accessory* option to add an accessory to the fixture.
- Once you hang a fixture on a pipe, you cannot drag it onto a mountable yoke.
- To remove a fixture from a mountable yoke, right-click the fixture, and then select *Remove Accessory*. In the resulting tree menu, you can either remove the accessory from the fixture, or remove the fixture from the mountable yoke.
- If you insert a fixture onto a yoke by using a shortcut, and then you insert another fixture onto a pipe using the same shortcut, when you right-click, the option *Finish Placing Accessories* appears (instead of *Finish Placing Fixtures*).
- When you insert a fixture on a mountable yoke in CAD mode, you can no longer focus the beam by dragging it in CAD mode. Additionally, the beam direction might not point straight down.

To insert an accessory

1. Open the Library Browser.
2. Click on the Accessory section tool.
3. Find the accessory that you want to insert.
4. Double-click the accessory name.
5. Click on the fixtures to insert the accessory.

Tips:

- To place an accessory on a selected fixture, right-click on the fixture, and then click **Add Accessory**. Use the browser to find the accessory you want to insert. Follow this procedure in Data mode and only those accessories that can be applied to the selected fixture will be available.
- To place an accessory on a group of selected fixtures, use the Accessory library. With the fixtures selected, find the accessory in the Library Browser. Right-click on the accessory name, and then click **Insert**.
- You can add the correct frame size to a fixture using Add Accessory.

To create a new shortcut to an accessory

Refer to “To create a new shortcut” on page 56.

Placing color and gobos

You can drop color and gobos onto fixtures by using the Library Browser or the Quick Tools method. Colors and gobos from many manufacturers are included in the library. You can preview the color or gobo by double-clicking on the name.

Note: For more information on Quick Tools, see “Quick Fixture tools” on page 299.

To insert colors and gobos using the Library Browser

1 Open the Library Browser.

2 Click the Color ( ) or Gobo ( ) section tool.

3 Find the color or gobo that you want to insert.

4 Right-click the color/gobo name, and then click **Insert**.

Result: The cursor shows a “Q”, the symbol denoting that Quick Tools have been enabled.

5 Click on the fixtures to which you want to add the color or gobo.

6 Right-click, and then select **Finish Quick Tools** to finish placing the color or gobo.

To insert colors and gobos using the Quick Tools

1 Click **Tools** > **Quick Tools**.

Result: The Quick Fixture Tool appears.

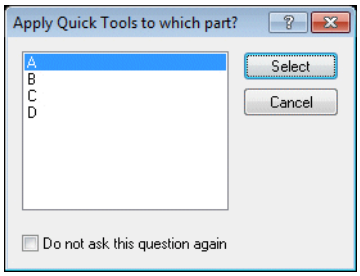
2 Under the Accessories heading at the bottom of the window, click the ellipsis button (...) beside the feature that you want to add (either color or gobo), and then choose the color or gobo in the resulting window.

3 Click *OK*.

Result: The cursor shows a “Q”, the symbol denoting that Quick Tools have been enabled.

4 Click on the fixtures to which you want to add the color or gobo.

Note: If you want to apply color to a multi-source light that has multiple circuits (such as the 528 Borderlight 6’ 4” Circuit), the following window appears, prompting you to select the circuit to which you want to apply the color:



In this case, select the circuit, and then click *Select*. To choose multiple circuits, press CTRL, and then select the desired circuits. Click *Select* when you are finished.

5 Right-click, and then select **Finish Quick Tools** to finish placing the color or gobo.

To create a new shortcut to a color or gobo

1 Click the *Col/Gobo* shortcut bar.

2 Right-click in the open space on the bar, and then choose **New Shortcut**.

Result: The *Color Select or Gobo Select* dialog box is displayed.

3 Select a color or gobo from the tree menu on the left side.

Result: The color or gobo is displayed on the right side.

4 Click *Select*.

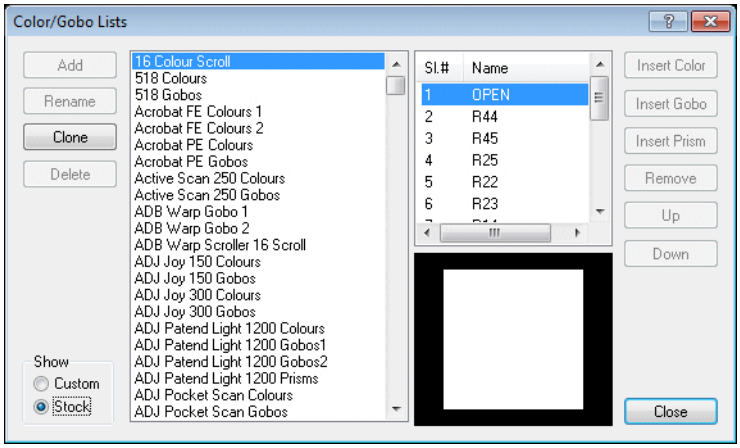
Color / gobo lists

Color and gobo lists are required for automated lighting fixtures and scrollers. By default, when inserted, automated fixtures and scrollers are set to use the manufacturer default color or gobo list. WYSIWYG contains a library of “stock” color and gobo lists provided by the fixture manufacturers. If you are using a custom set of colors or gobos, or want to generate a scroll for a color scroller, you will need to create a custom color or gobo list.

Note: To create a custom color or gobo list for a fixture or scroller, it is strongly recommended that you Clone the existing list for that fixture or scroller, modify the clone, and then insert that clone into the fixture, as described in “To insert custom scrolls into scrollers” on page 319.

To view an existing library list

1 From the **Library** menu, choose **Edit Color/Gobo Lists**.



2 In the *Show* section, click *Stock*.

3 Scroll to find the list you want to view.

4 Click on the list name. A list of slots and the color or gobo for each slot is displayed in the slot window. If *OPEN* appears in the Name column for a slot, the color, gobo, or prism has not been assigned or is intentionally left *OPEN*.

5 Click on a slot to see the preview of that color or gobo in the box below the slot window.

6 Click *Close* when you are done.

To create a custom color / gobo list

1 From the **Library** menu, choose **Edit Color/Gobo Lists**.

2 In the *Show* section, click *Custom*.

3 Click *Add*.

4 Type the name of the new list.

5 Type the number of slots on the wheel or scroll.

6 Click *OK*.

7 Click on the slot you wish to edit.

8 Click *Insert Color*, or *Insert Gobo*, or *Insert Prism*. Make sure if you are inserting different types of media into the same list that this is actually possible for that fixture or accessory.

9 Click on the item you want to insert, and then click *Select*.

10 Repeat for the remaining slots.

11 To reorder the items in the slots, click on the slot, and then click *Up* or *Down* to move that item to a new position.

12 To erase a color or gobo selection from a slot, click *Remove*. The slot returns to the default selection of *OPEN*.

13 Click *Close* when you are done creating lists.

Important: It is recommended to clone the stock wheel or list associated with the fixture(s) and edit the cloned list. The cloned list inherits the properties of the original list.

To clone a stock color / gobo list

1 From the **Library** menu, choose **Edit Color/Gobo Lists**.

2 In the *Show* section, click *Stock*.

3 Scroll to find the list you want to clone.

4 Click *Clone*.

5 Type a name for the cloned list, and then click *OK*.

Result: The cloned list appears in the custom lists.

To rename a color / gobo list

1 From the **Library** menu, choose **Edit Color/Gobo Lists**.

2 In the *Show* section, click *Custom*.

3 Select the color/gobo list that you want to rename.

4 Click *Rename*.

5 Type the new name for the list, and then click OK.

Result: The list is renamed.

To delete a color / gobo list

1 From the **Library** menu, choose **Edit Color/Gobo Lists**.

2 In the *Show* section, click *Custom*.

3 Select the color/gobo list that you want to remove.

4 Click *Delete*.

Result: The list is removed from the color/gobo list.

To insert custom scrolls into scrollers

1. Select the fixtures with scrollers for which you want to assign a custom scroll.
2. From the **Edit** menu, choose **Object Properties**.
3. Select the *Fixture* tab.
4. In the fixture list, highlight the scroller.
5. Select the *Scroll* tab.
6. Select the desired list from the pull-down list.

   Note: Only custom lists that have the appropriate number of slots and media type are available.
7. Click *OK*.

To insert custom wheels into automated fixtures

1. Select the fixtures for which you want to assign a custom wheel.
2. From the **Edit** menu, choose **Object Properties**.
3. Select the *Fixture* tab.
4. Select the appropriate tab (for example, Color 1, Gobo, <Gobo>).
5. From the drop-down list, select the desired list.

   Note: Only custom lists that have the appropriate number of slots and media type are available.
6. Click *OK*.

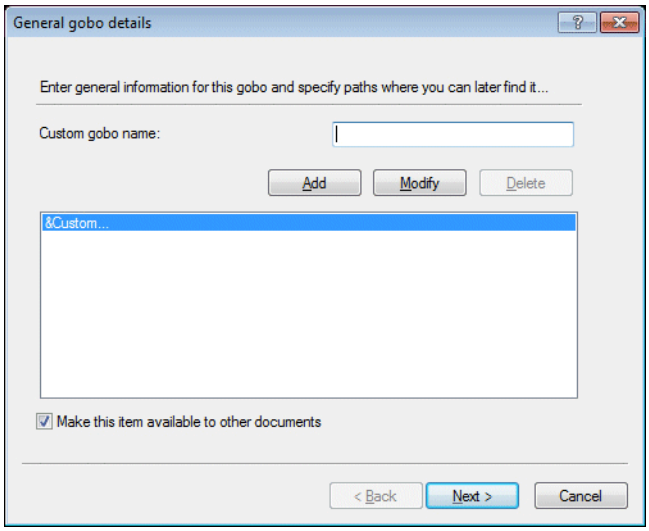
Custom gobos

You can generate custom gobos if you have the artwork in either bitmap or jpeg format. The Gobo Wizard walks you through the steps of making a custom gobo. Custom gobos are stored in the gobo library under the Custom category.

Custom gobos require an association with an existing WYSIWYG gobo to be visible in wireframe views. The associated gobo is used in lieu of the bitmap or jpeg image. In shaded views and renderings, only the custom gobo footprint is visible. Custom gobos do not break up the cone of light.

To create a custom gobo

1 From the **Library** menu, choose **Gobo Wizard**.



2 Type the custom gobo name.

3 Add or modify the path.

Note: The path is the location in the library where the gobo will be stored. A gobo assigned to the path “Custom/Corporate_Logos” would be available in the custom section of the gobo library in a sub-category named Corporate Logos.

4 To make this gobo available to other documents, check the appropriate box.

5 Click *Next*.

6 Type the path to the custom artwork file or use the browser [...] to open the image. The image file must be in either bitmap or jpeg format.

Result: A preview of the image appears in the pane on the right and the image’s attributes are recorded in the box on the left.

7 Click *Next*.

Result: A short description of the next step appears. You will need to choose an existing library gobo to be displayed when the custom artwork cannot be displayed.

8 Click *Next*.

9 Navigate the library to find a suitable association gobo.

Note: Generic association gobos are stored in the Proxy category.

10 Select the desired gobo.

11 Click *Finish*.

Result: The custom gobo is placed in the gobo library as per the path specified in step 3.

To delete custom gobos

If you do not want to keep a custom gobo any longer, you can delete it from the library browser.

Notes:

- You cannot delete a custom gobo if it is currently inserted in your plot; instead, you must delete the item from your plot, save and close the file. When you reopen the file you can delete the item from the library browser.

- When you delete a custom gobo, you do not delete the shortcut associated with it. You must delete shortcuts manually.

1. In the Library Browser, navigate to the folder where you stored the custom library item.
2. Highlight the library item name, and then right-click and choose **Delete**.

   Note: This option is disabled if the item is currently inserted in your plot. You must delete the item from your plot, and then save and close the file. When you reopen the file you can delete the item from the library browser.
3. A message box appears telling you the action is irreversible. Click *Yes* to delete the item.

LED Wizard

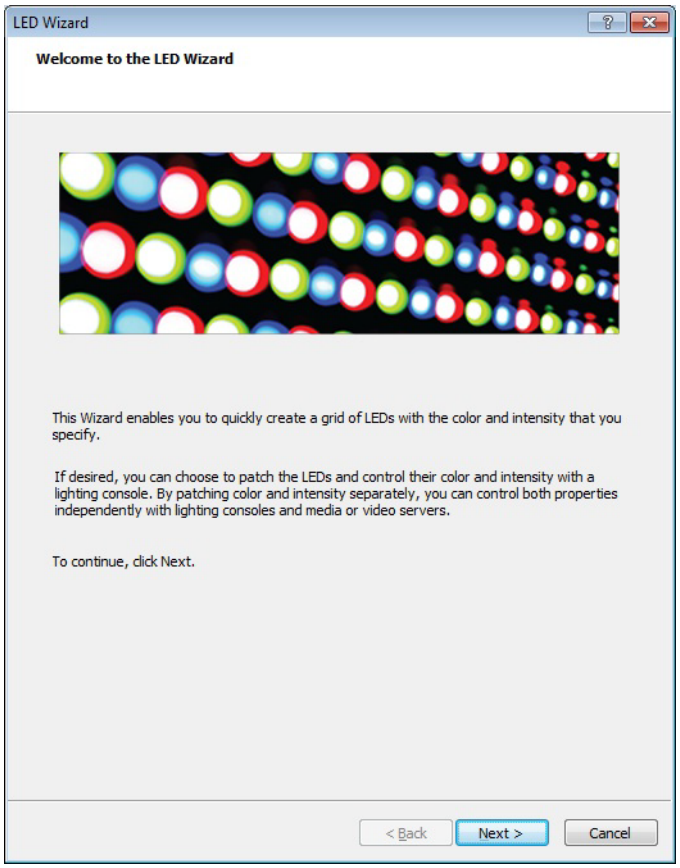
Introduction

This Wizard enables you to quickly create a grid of LEDs that simulates LED walls or panels, as well as LED webs, star cloths and video curtains. The grid can have Image or Video Sources applied to it, or can be patched to DMX universes for control via LED mapping software or lighting consoles.

To use the LED Wizard

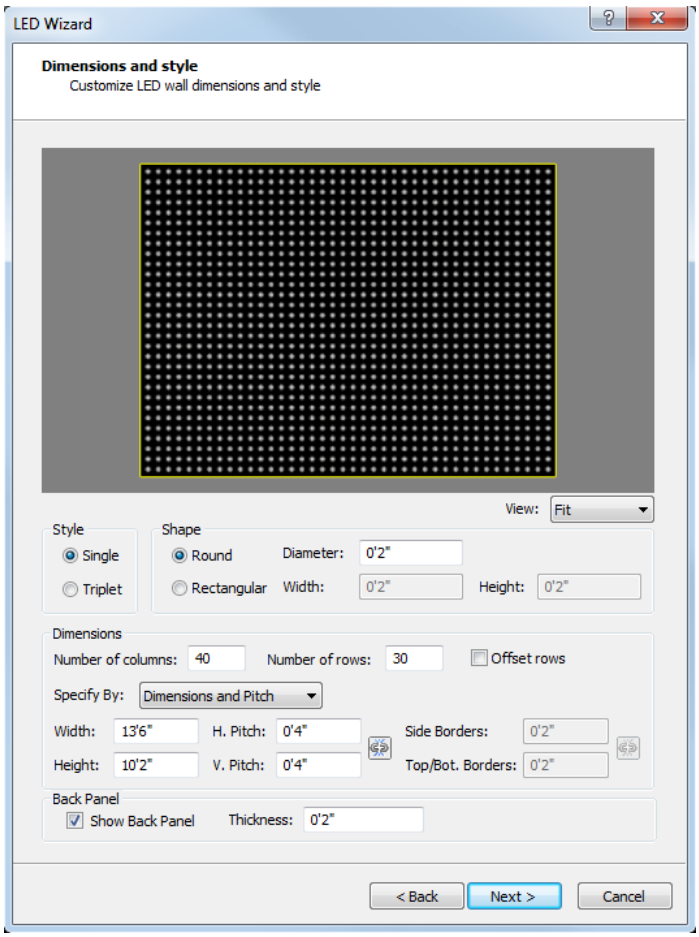
1 From the **Draw** menu, choose **LED Wizard**.

Result: The LED Wizard appears.



2 Click *Next*.

Result: The Dimensions and style window appears.



3 Use the options in the View drop-down list to control the "zoom level" of the preview area:

- *Fit*: Choose this option to fit the grid to the preview area.
- *Zoom*: Choose this option to zoom in on the center of the grid, to check the image or video that was mapped to the pixels.
- *Corner*: Choose this option to zoom in on the lower-left corner of the grid. This is useful for checking how the frame appears in relation to the rest of the grid.

Note: The preview of the LED device will update as you provide its style and dimensions values. This helps you visualize your LED panel or wall before you move onto the next step.

4 In the Style section, choose the style of LEDs that you want to use in the grid you are creating:

- *Single*: Choose this option to use single LEDs which produce (all) three colors—Red and Green and Blue.
- *Triplet*: Choose this option to use LED clusters comprised of three LEDs, each of which produces a single color—Red or Green or Blue).

5 In the Shape section, choose the shape of the individual LEDs to use in the grid, and then specify their size:

- *Round*: Click this option if you want the Single LED or each element of a Triplet to be round, and then type the diameter of the LEDs in this grid.

- *Rectangular*: Click this option if you want the Single LED or each element of a Triplet to be rectangular, and then type the width and height of the LEDs in this grid.

6 In the Dimensions section, define the layout of the grid by specifying the number of rows and columns of LEDs:

- *Number of columns*: Type the number of columns in the grid.
- *Number of rows*: Type the number of rows in the grid.
- *Offset rows*: Click this checkbox to offset the rows in the grid by one-half of the Horizontal Pitch (described below).

7 Use the *Specify By* drop-down and the fields below it to select the parameters of the device (i.e., LED panel, LED wall, LED web, video wall, etc.) that you will use to define this LED grid:

- Choose **Dimensions and Pitch** to define the grid's height and width and pixel pitch. (The pitch is the center-to-center distance between LEDs in the grid. The height and width are the overall height and width of the device as seen from the front—i.e., the height and width of an LED panel or LED wall.) When you choose this option, the LED Wizard will calculate the width of the device's Borders for you.
  - *Width*: Type the overall width of the device being simulated.
  - *Height*: Type the overall height of the device being simulated.
  - *H. Pitch*: Type the horizontal center-to-center distance between LEDs in the grid of the device being simulated.
  - *V. Pitch*: Type the vertical center-to-center distance between LEDs in the grid of the device being simulated.

  Tip: Since, in many cases, LED panels or walls have the same horizontal and vertical pixel pitch, you may click the lock button to apply the same value to both.

- Choose **Dimensions and Borders** to define the height and width of the device being simulated, and its borders' horizontal and vertical sizes. (The height and width are the overall height and width of the device as seen from the front—i.e., the height and width of an LED panel or LED wall. The border is the distance between the edge of the device and the edge of the first row and/or column of LEDs in the grid.) When you choose this option, the LED Wizard will calculate the grid's pixel pitch for you.
  - *Width*: Type the overall width of the device being simulated.
  - *Height*: Type the overall height of the device being simulated.
  - *Side Borders*: Type the distance between the left edge of the device being simulated and the edge of the left-most column of LEDs. The same value will be applied to the right side of the device, between its right edge and the edge of the right-most row of LEDs.
  - *Top/Bottom Borders*: Type the distance between the top *edge* of the device being simulated and the *edge* of the top row of LEDs. The same value will be applied to the bottom of the device, between its bottom *edge* and the *edge* of the bottom row of LEDs.

  Tip: Since, in many cases, devices such as LED panels or walls have the same-size horizontal and vertical borders, you may click the lock button to apply the same value to both.

- Choose **Pitch and Borders** to define the pixel pitch and borders' horizontal and vertical sizes of the device being simulated. (The pixel pitch is the center-to-center distance between LEDs in the grid. The border is the distance between the edge of the device and the edge of the first row and/or column of

LEDs in the grid.) When you choose this option, the LED Wizard will calculate the device's height and width for you.

- *H. Pitch*: Type the horizontal center-to-center distance between LEDs in the grid of the device being simulated.
- *V. Pitch*: Type the vertical center-to-center distance between LEDs in the grid of the device being simulated.
- *Side Borders*: Type the distance between the left edge of the device being simulated and the edge of the left-most column of LEDs. The same value will be applied to the right side of the device, between its right edge and the edge of the right-most row of LEDs.
- *Top/Bottom Borders*: Type the distance between the top *edge* of the device being simulated and the *edge* of the top row of LEDs. The same value will be applied to the bottom of the device, between its bottom *edge* and the *edge* of the bottom row of LEDs.

Tip: Since, in many cases, LED panels or walls have the same-size horizontal and vertical pixel pitch as well as borders, you may click the lock button to apply the same value to both.

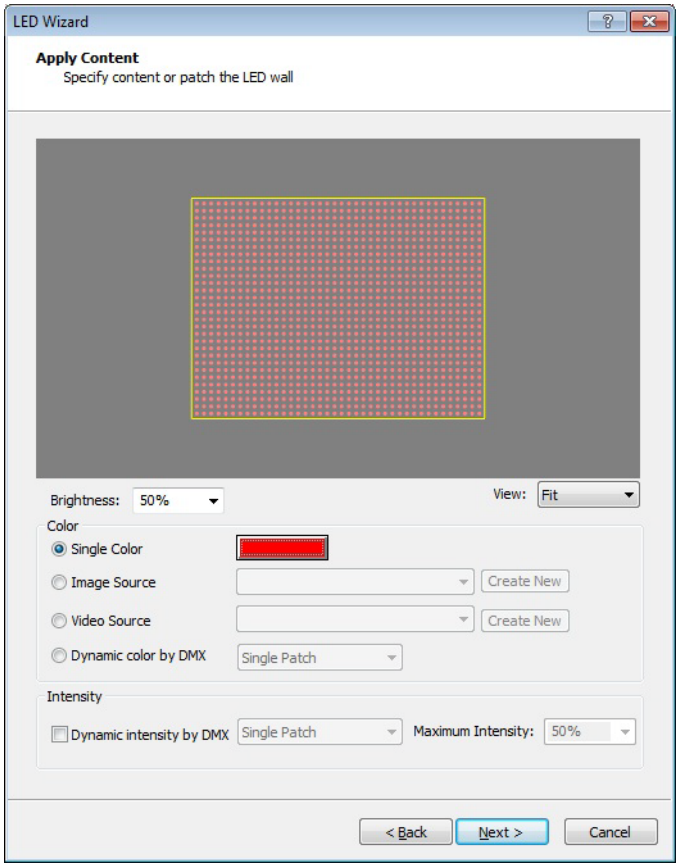
8 In the Back Panel section, choose the back panel settings for LED Wall.

- *Show Back Panel*: Enable to create a black back panel for the LED Wall. This option is enabled by default.
- *Thickness*: Type in the thickness of the back panel.

Tip: After the LED Wall is inserted the file, the Back Panel settings can still be changed from the properties menu of the LED Wall.

9 Click *Next*.

Result: The Apply Content window appears.



10 Use the *Brightness* drop-down to set the overall brightness of the LED grid in CAD and Design modes.

Note: If you do not enable Dynamic Intensity by DMX, the static intensity that you define will also apply in Live mode.

11 In the Color section, select what this LED grid will display:

- *Single Color*: Choose this option in order to have the grid display the selected color in all modes; click the color box next to this option to select the color to be displayed.
- *Image Source*: Choose this option in order to have the grid display the image from an Image Source or Subsource that you created in the Image Manager, and then select the Image Source or Subsource to display from the drop-down. (For more information on using the Image Manager, see “Image Manager” on page 343.) Click *Create New* to launch the Image Manager to create a new image source or subsource.

Tip: For optimum results, the number of rows and columns of this LED grid should be proportional to the dimensions of the Image Source or Subsource being selected here.

- *Video Source*: Choose this option in order to have the grid display the video from a Video Source or Subsource that you created in the Video Manager, and then select the Video Source or Subsource to display from the drop-down. (For more information on using the Video Manager, see “Video Manager” on page 335.) Click *Create New* to launch the Video Manager to create a new Video Source or Subsource.

IMPORTANT: When a video is used on an LED grid, if that video is not playing, that grid will not appear in Shaded Views in Design and Live modes. To play the video, use the Video Designer Tool in Design or Live mode.

Tip: For optimum results, the number of rows and columns of this grid should be proportional to the dimensions of the Video Source or Subsource being selected here.

- *Dynamic Color by DMX*: Choose this option to control the color of the LEDs in this grid via DMX (by connecting to a lighting console or to LED mapping software), and then select how the color channels should be patched:
  - *Single Patch*: Select this option from the drop-down in order to control the color of this entire grid with a single set of DMX channels. After selecting this option, if you choose red on your console, every point source in the entire grid will be red.
  - *Multi Patch*: Select this option from the drop-down to control individual LEDs in this grid via separate sets of DMX channels. After selecting this option, you will have the ability to control the color mix for every LED in the grid individually.

Tip: Select this option when controlling the LED grid from specialized LED mapping software.

Note: If this option is selected, the LEDs will appear in the color of the Layer on which this grid is located in all Shaded views, except those in Live mode (where they will appear in the color provided by the incoming DMX data).

12 In the Intensity section, you have the option to select whether or not you wish to control the intensity of the grid dynamically, via DMX for Live mode.

Note: This option applies to Live mode only. The Brightness value will set the intensity if the LED grid for all other modes.

- *Single Patch*: Select this option from the drop-down in order to control the intensity of the entire grid with a single DMX channel.

- *Multi Patch*: Select this option from the drop-down in order to individually control the intensity of each LED in this grid via individual DMX channels.
- *Maximum Intensity*: Select the maximum intensity value of the LEDs in this grid when connected to a console and viewed in Live mode.

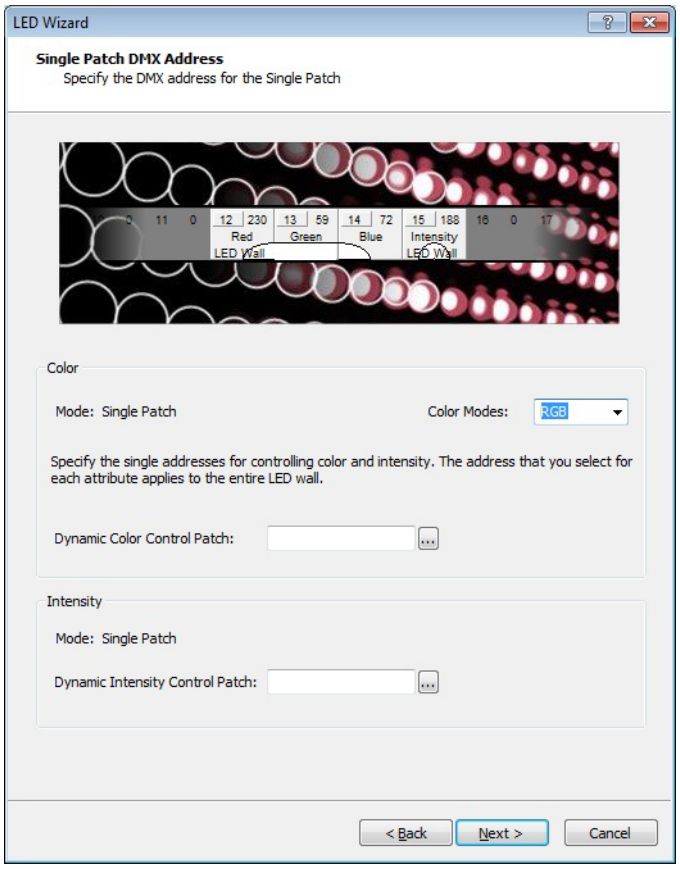
IMPORTANT: This value applies to the maximum intensity of the LEDs in Live mode. This is intended to control maximum brightness of each LED grid, because some LEDs are brighter than others. For example, if you set one LED grid to 100% and another LED grid to 50%, when you then control them via DMX in Live mode and set the console channel to full (255), the LED grid with Maximum Intensity set to 100% will be brighter than the one set to 50%.

13 Click *Next*.

Result: If the content of your LED grid is controlled by a Single Color, Image Source or Video Source, you are Finished. Otherwise, you've selected to control color and/or intensity dynamically, and the LED Wizard proceeds to the appropriate patching page, as shown below. Locate your patching scenario, and follow the steps listed.

Single Patch DMX Address

This page is displayed if you set "Dynamic Color by DMX" to the Single Patch option. If you also set "Dynamic Intensity by DMX" to the Single Patch option, it will be active, otherwise it will disabled.



a. Select one of several Color Modes by which color is controlled, enter the DMX control channels for the *entire* grid, and then click *Next*:

- *Color Modes*: Choose between RGB, RGBA, RGBW and RGBAW, depending on the device you are simulating. "RGB" stands for "Red, Green, Blue"; "A" stands for "Amber"; "W" stands for "White". The Color Mode setting

defines the number of DMX channels required to control the color mix for this *entire* grid.

Note: WYSIWYG does not actually show white or amber LEDs in its simulation. The additional “A” and/or “W” channels are offered so this Wizard can account for these channels in the patch, if necessary.

- *Dynamic Color Control Patch*: Type the universe and starting channel number for this patch, in the format Universe_Name.DMX_Channel_Number. (If the Universe_Name you typed does not exist, you will be asked if you wish to create it when you click *Next*.) Alternately, you may click the ellipsis button to manually select a previously-created universe (or create a new one), and then type the channel number.

Tip: Once you have finished creating this grid and have inserted it, switch to Data mode > Patch view, and click the universe to which you patched it, in order to verify that the patch has been applied correctly. For example, if you chose the RGBAW color mode, this grid should span five DMX channels, starting with the channel number you defined. (For more information on using the Patch view in Data mode, see “Working in the patch view” on page 366.)

- (If active) *Dynamic Intensity Control Patch*: Type the universe and channel number for this patch, in the format Universe_Name.DMX_Channel_Number. (If the Universe_Name you typed does not exist, you will be asked if you wish to create it when you click Next.) Alternately, you may click the ellipsis button to manually select a previously-created universe (or create a new one), and then type the channel number.

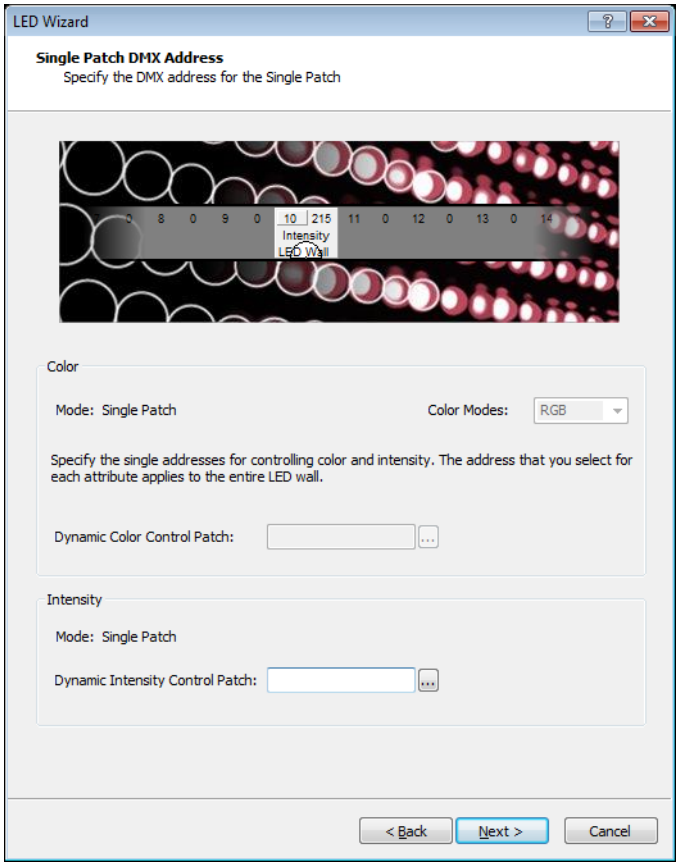
b. Click *Next*.

Result: The Finished page appears.

c. Proceed to step 13.

Single Patch DMX Address with Single Color, Image or Video Source

This page is displayed if you set “Dynamic Intensity by DMX” to the Single Patch option, and color is controlled by Single Color, Image Source or Video Source.



a. Enter the address of the (single) DMX channel used to control the intensity of this entire grid, then click *Next*:

- *Dynamic Intensity Control Patch*: Type the universe name and channel number for this patch, in the format Universe_Name.DMX_Channel_Number. (If the Universe_Name you typed does not exist, you will be asked if you wish to create it when you click Next.) Alternately, you may click the ellipsis button to manually select a previously-created universe (or create a new one) and type the channel number.

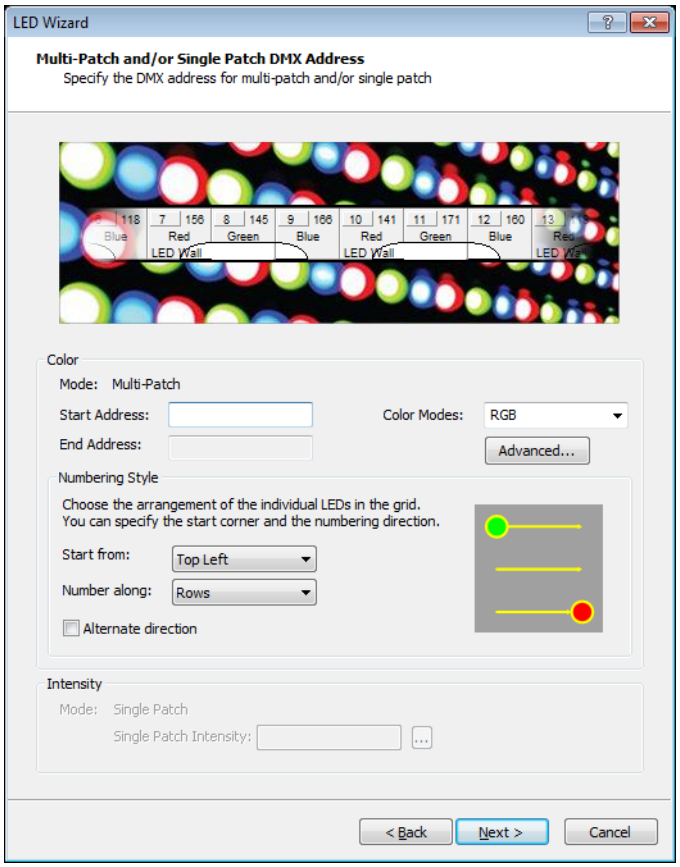
b. Click *Next*.

Result: The Finished page appears.

c. Proceed to step 13.

Multi Patch and/or Single Patch DMX Address

This page is displayed if you set “Dynamic Color by DMX” to the Multi Patch option. Then, depending on the choice for “Dynamic Intensity by DMX”, it will be in Single Patch mode or Multi Patch mode accordingly.



a. In the Color section, enter the DMX address at which the “first” LED in the grid is to be patched, define which LED in the grid is “first” and how the rest of the LEDs “follow”, select one of several Color Modes by which color is controlled (for the entire grid), and then click *Next*.

- *Start Address*: Type the universe and channel number for the first LED in the grid, in the format Universe_Name.DMX_Channel_Number. (If the Universe_Name you typed does not exist, you will be asked if you wish to create it when you click *Next*.) The channel numbers for the rest of the LEDs in this grid will be “calculated” automatically by the LED Wizard, and the last channel number (and universe) will be displayed in the End Address field.

Note: In cases where the number of DMX channels required to patch the grid in this manner exceed 512 (i.e., the maximum number of channels in a DMX universe) the LED Wizard will automatically create new universes that start with the same name and have a number (starting with “1” and incrementing from there) appended to the name. For example, if the first LED was patched to universe ‘LED’, but two more universes were needed to patch the grid, the LED Wizard would create two new universes named ‘LED1’ and ‘LED2’.

- In the Numbering Style subsection, define which LED in the grid is to be considered “first” using the *Start from* drop-down, and how the rest of the LEDs are to follow it using the *Number along* drop-down. Click the *Alternate direction* checkbox to indicate that the numbering direction should change with every row or column.
- *Color Modes*: Choose between RGB, RGBA, RGBW and RGBAW, depending on the device you are simulating. “RGB” stands for “Red, Green, Blue”; “A”

stands for “Amber”; “W” stands for “White”. The Color Mode setting defines the number of DMX channels required to control the color mix for each LED in this grid.

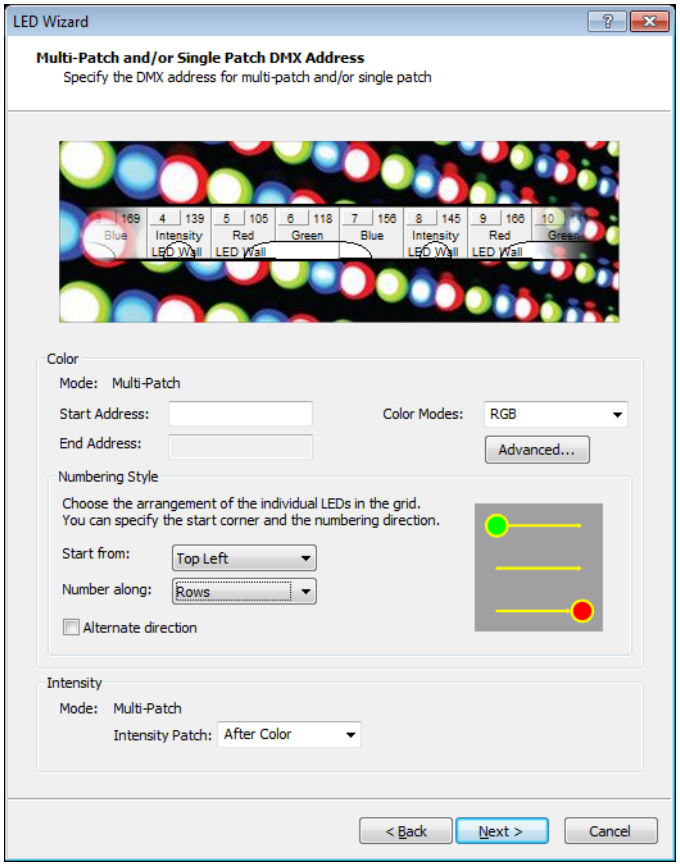
Note: WYSIWYG does not actually show white or amber LEDs in its simulation. The additional “A” and/or “W” channels are offered so this Wizard can account for these channels in the patch if necessary.

a. In the Intensity section, specify the patching information for intensity.

- *Single Patch Intensity*: Type the universe and channel number for this patch, in the format Universe_Name.DMX_Channel_Number. (If the Universe_Name you typed does not exist, you will be asked if you wish to create it when you click Next.) Alternately, you may click the ellipsis button to manually select a previously-created universe (or create a new one) and type the channel number.

Note: If the Intensity mode is set to Multi-Patch, then the following page appears. In this case, the intensity channel controlling the individual LED will either be before or after the color channels.

- *Intensity Patch*: If the intensity channel should follow the color channels, then set the drop-down to “After Color”. Otherwise, if the intensity channel is before the color channels, then set the drop-down to “Before Color”.



Note: For tips on advanced patching techniques, see “Advanced patching” on page 333.

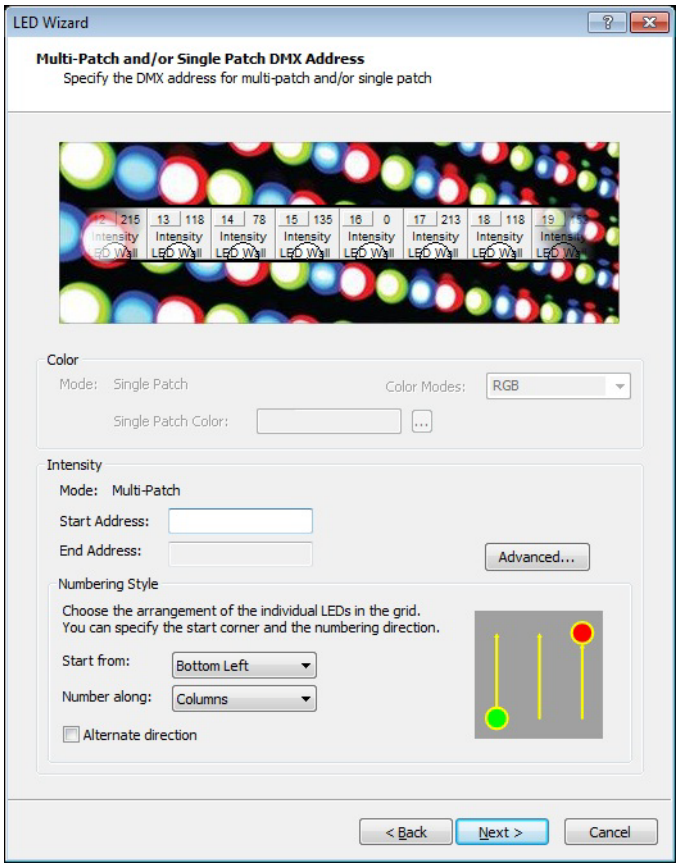
b. Click *Next*.

Result: The Finished page appears.

c. Proceed to step 13.

Multi Patch and/or Single Patch with Color Set to Single Patch and Intensity Set to Multi-Patch

This page is displayed if you set “Dynamic Color by DMX” to the Single Patch option, and the “Dynamic Intensity by DMX” is set to Multi Patch.



a. Enter the DMX address at which the “first” LED in the grid is to be patched, define which LED in the grid is actually “first” and how the rest of the LEDs “follow”, then click Next:

- *Start Address:* Type the universe and starting channel number for the first LED in the grid, in the format Universe_Name.DMX_Channel_Number. (If the Universe_Name you typed does not exist, you will be asked if you wish to create it when you click Next.) The channel numbers for the rest of the LEDs in this grid will be “calculated” automatically by the LED Wizard, and the last channel number (and universe) will be displayed in the End Address field.

Note: In cases where the number of DMX channels required to patch the grid in this manner exceed 512 (i.e., the maximum number of channels in a DMX universe) the LED Wizard will automatically create new universes that start with the same name and have a number (starting with “1” and incrementing from there) appended to the name. For example, if the first LED was patched to universe ‘LED’, but two more universes were needed to patch the grid, the LED Wizard would create two new universes named ‘LED1’ and ‘LED2’.

- In the Numbering Style subsection, define which LED in the grid is to be considered “first” using the *Start from* drop-down, and how the rest of the LEDs are to follow it using the *Number along* drop-down. Click the *Alternate direction* checkbox to indicate that the numbering direction should change with every row or column.

Note: For tips on advanced patching techniques, see “Advanced patching” on page 333.

b. Click Next.

Result: The Finished page appears.

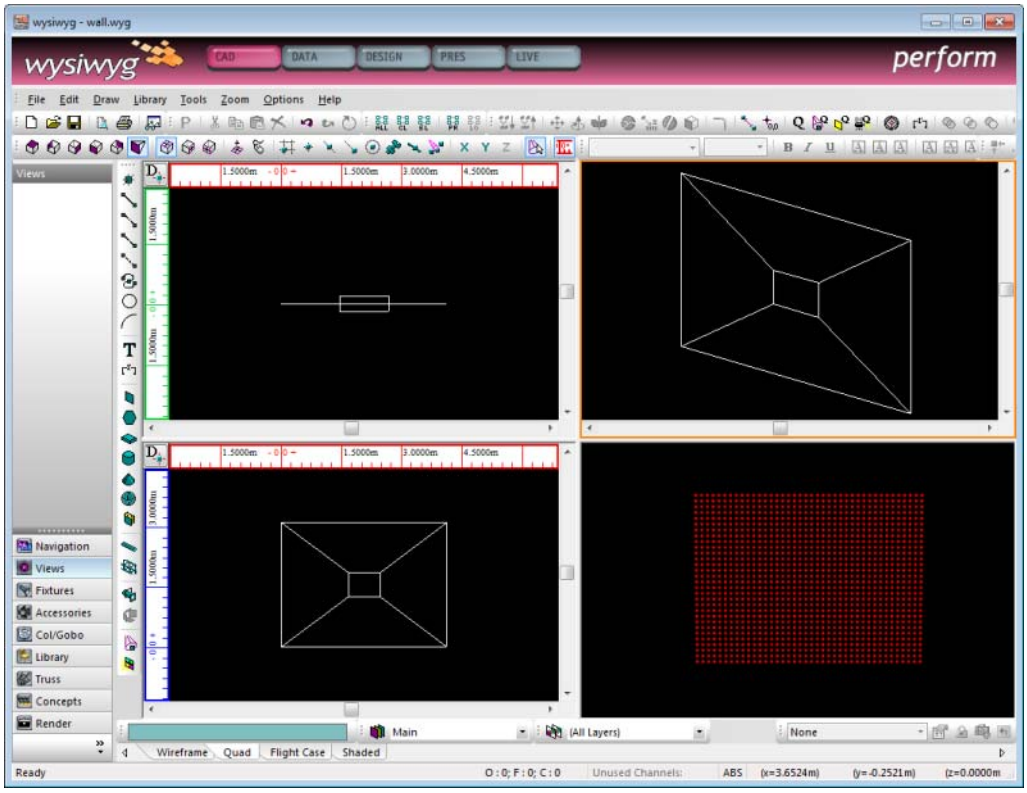
c. Proceed to step 13.

14 Click *Finish*.

Result: The LED grid attaches to your cursor.

15 Click to place the grid into your drawing at the desired location, or type in the coordinates at which to place it.

Result: The LED grid is placed into the drawing.



Advanced patching

This option offers more control in defining your LED grid's patching information. By using Auto Insert, each universe will start at channel 1 and then automatically create all the universes required to control your LED grid for you. You then have the option to select a row and edit the start and end channels for each universe if you wish to do so. Otherwise, you can manually create each universe and specify start and end channels for each.

Notes:

- Before typing addresses, consult the figures shown under *Addresses required* and *Addresses in pool* for an up-to-date listing of the number that you will need for your grid and the number that you have already provided.
- Create the patch domain in the address pool box, following the format DomainName.StartAddress-EndAddress (for example, **a.1-169**).
- You can type multiple patch domains (for example, **a.1-160** and **b.1-20**). If you assign more domains than are required, the extra domains are ignored.
- Use the arrow buttons to adjust the order of the patches (the order in which they appear is the order in which they are populated).

- If you require more than 512 addresses (the standard DMX universe), then you must add additional patch domains to meet your requirement. The addresses can be split over universes.

Type the addresses manually

a. In the box provided, type the required addresses in the format DomainName.StartAddress-EndAddress (for example, **a.1-169**).

b. Use the arrow buttons to adjust the order of the patches (the order in which they appear is the order in which they are populated).

c. Click *Next*, and then proceed to the next step.

Use Auto Insert to create the addresses automatically

a. Click the *Auto Insert* icon to have WYSIWYG automatically create all the addresses that you need for the grid. In the resulting Auto Insert window, type the Patch Domain Prefix (for example, **a**).

b. In the *Number of Channels Per Domain* box, accept the default value or type a new value based on the number of channels that you can control.

c. In the *Start Address* box, type the start address for the patch domain.

d. Click *OK* to have WYSIWYG populate the End Address (based on the number of channels required).

e. Click *Next*, and then proceed to the next step.

Select the addresses from a list

a. If you do not want to manually type the addresses, click the *New* icon, and then click ellipses button (...).

- To create a new patch domain, click *Create New*, and then type the domain in the *Domain Name* box (for example, **a**).
- To choose a patch domain from those that you have saved with your current WYSIWYG file, click *Select From List*, and then highlight the domain name.

b. Type the Start and End addresses for this patch universe.

c. Click *OK*.

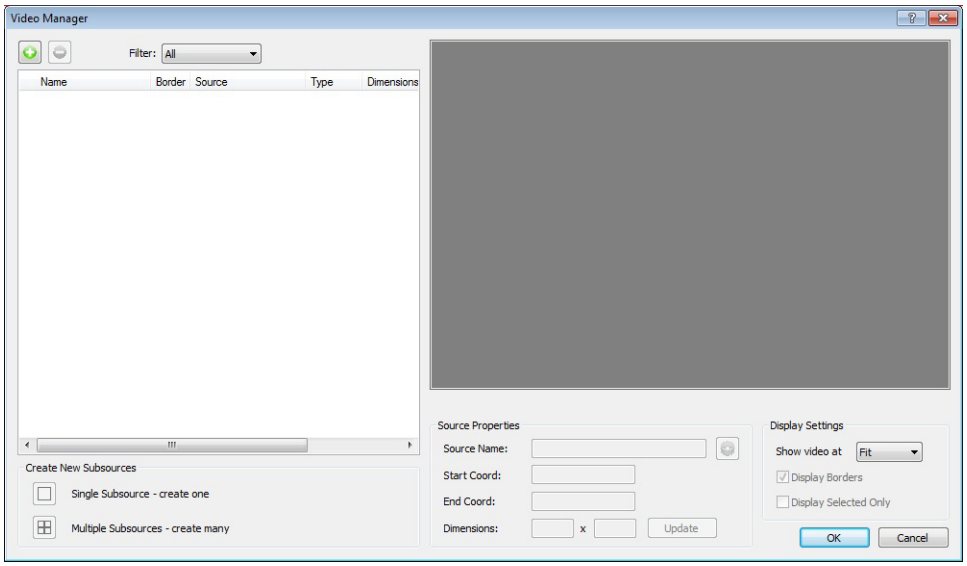
d. Click *Next*, and then proceed to the next step.

Video Manager

Introduction

You can bring a live or pre-recorded video into WYSIWYG and play it back while you set looks in Design mode and while you precue in Live mode.

The Video Manager enables you to create video sources and subsources (and even split these subsources further), which you can then assign to screens, 3D primitives (surfaces, risers, spheres, cylinders, and cones), LED Walls, and to projectors that you have inserted in your file from the Video Projection section of the Fixtures library.



Live video comes from an external source, such as a video capture device or a CITP/MSEX video stream from a media server. Pre-recorded video comes from video files, either in AVI or MPG format.

Notes: If you cannot view these file types, you may not have the proper CODEC installed on your PC. Install the appropriate CODEC, and then try viewing the file again.

Video sources can be split it up into subsources. This enables you to apply portions of the video to multiple screens, 3D primitives, LED Walls, or projectors. For details, see “Splitting video into subsources” on page 338.

To play the video, you use the Video Designer Tool in Design mode. For more information, see “To use the Video tool” on page 480. You can view the video in any of the shaded views, using the controls on the Video designer tool to pause, fast forward, rewind, or stop the video.

You can also patch the *control* of the video source to a console. This means that you can use the console to control the progress of a live video stream or a video file by making the video play, pause, fast-forward, or rewind; you cannot use WYSIWYG to change different aspects of the video itself.

To control the video source, the DMX patch is given three channels. For details on controlling the video with the console device in Live mode, see “To control a DMX patched video source with a console” on page 531.

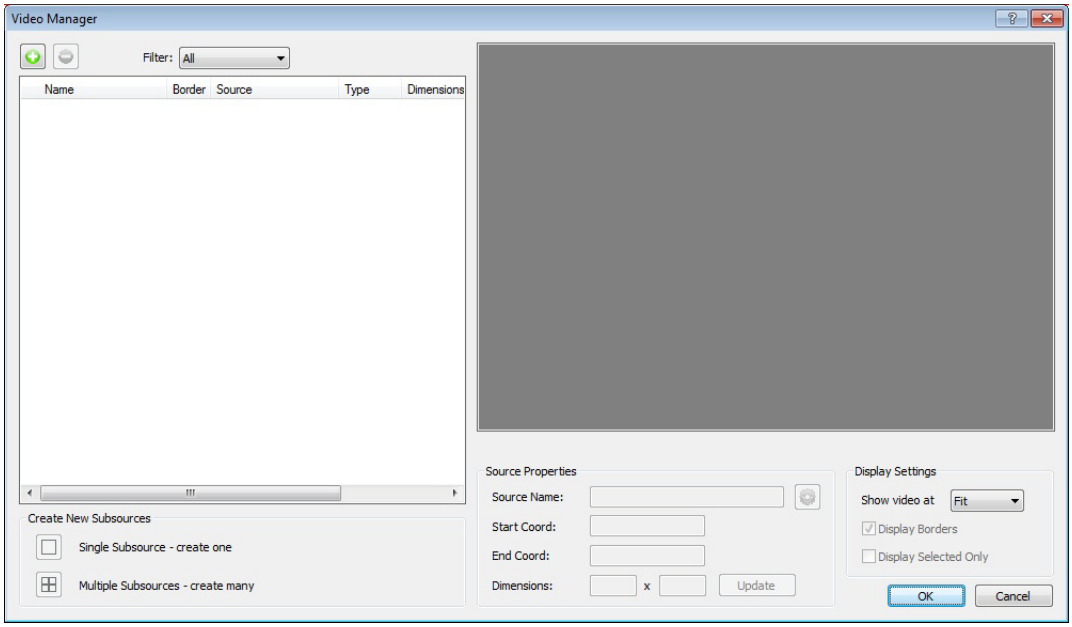
Notes:

- Before you render your drawing, you can either pause the video at the precise image you want to see in the final rendering, or you can let the video run while the Render Wizard processes the information in your drawing. In this case, the Render Wizard captures the video frame that was showing when it processed the screen information.
- If the video control is patched and DMX connected in Live Mode, then you will not be able to control the video using any of the Video Designer Tool commands in Design mode. Video sources can only be controlled by a designer tool when the DMX source is disconnected; therefore, you must first disconnect the applicable console from Live Mode before using any of the Video Designer Tool commands.
- For details on drawing screens, see “Drawing Camera paths” on page 167.

To access the Video Manager

1 In CAD mode, select **Tools** > **Video Manager**.

Result: The Video Manager appears.

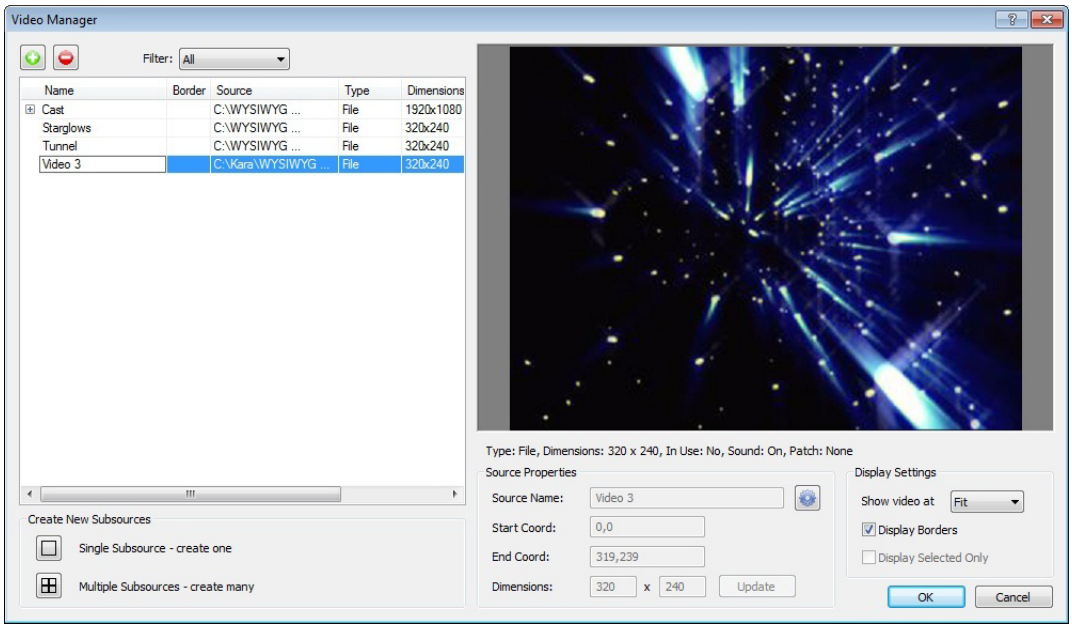


To create a new video source in the Video Manager

1 In the Video Manager, click the *New* icon.

2 Browse to locate the video on your computer, and then click *OK*.

Result: The Video is added as a source.



Notes:

- *Name*: The video's file name.
- *Border*: When there are subsources, the Border columns shows the border color. To change the border color, click in the box and choose the new color from the color picker that appears.
- *Source*: The path to the physical file.
- *Dimensions*: The file's dimensions in pixels.
- *In Use*: Informs you of whether the video source is in use. If the video source is currently in use, it cannot be deleted from the Video Manager.
- *Mute*: Indicates if the sound has been muted in the video.
- *Patch*: Indicates if the video source has been patched.
- Display Settings
  - *Show video at*: Select the zoom level for the video. You can choose to "fit" the video in the display window, or show it at a percentage of its actual size: 100%, 200% or 300%.
  - *Display Borders*: Select this option to show the subsource borders in the display.
  - *Display Selected Only*: When you have selected a subsource in the table, click this check box to *only* show the subsource in the display area; if you do not select this checkbox when a subsource is selected, the entire source is shown (with the subsource borders highlighted in the display if you selected *Display Borders*).
- For details on patching the video control in Data mode, see "To patch the control of a video source" on page 369.
- For details on using the Video Designer Tool to play the video in Design mode, see "To use the Video tool" on page 480.
- For details on using a console device to control the video in Live mode, see "To control a DMX patched video source with a console" on page 531.

Splitting video into subsources

After you set up the video source in the Video Manager (see “To configure a new video source for streaming video” on page 341), you can take the video source (video file, capture card, or CITP) and literally split it up into subsources that you can apply to multiple screens, 3D primitives, LED Walls, or projector-style fixtures that you have inserted from the Video Projection section of the Fixtures library.

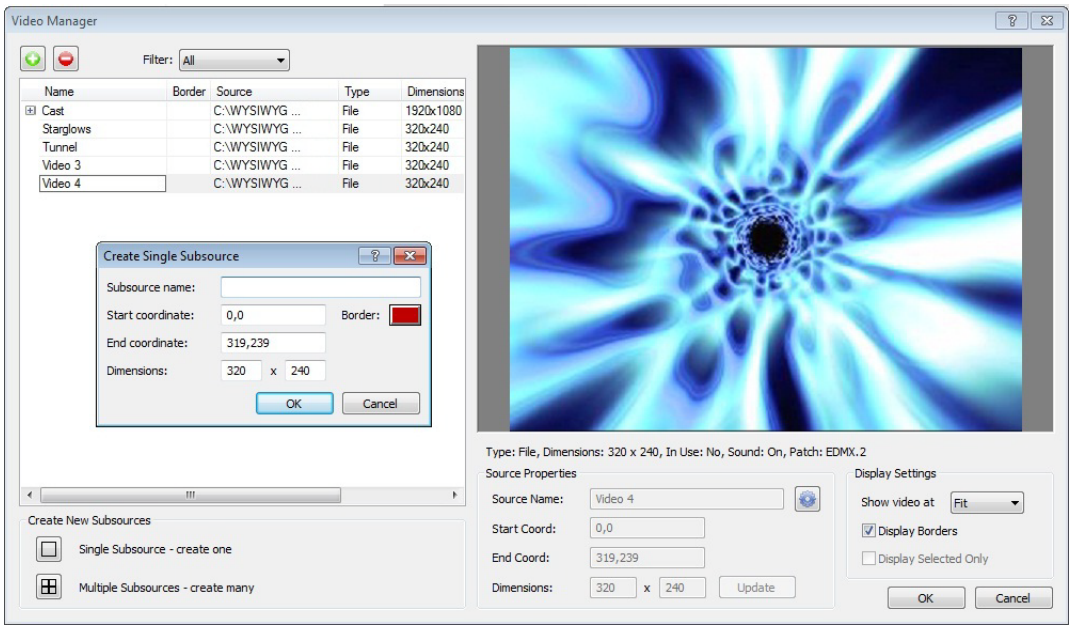
To do so, you click the *Split Video* tab in the Video Manager, select a video source that you set up, and then split it either manually (by defining the pixel coordinates of each split) or automatically/evenly (by setting the number of rows and columns). You can then split these subsources further, if desired.

Once you split the video into subsources, you can attach the subsources to the objects that you want to use to display the video, just as you would attach a regular video source, and then play the video using the Video Designer Tool (Design or Live modes) or via DMX control.

To create a single video subsource

1 In the left pane of the Video Manager, click to highlight the image for which you want to create a subsource.

2 In the bottom left corner of the window, click *Single Subsource - create one*.



3 Type the subsource name.

4 By default, the system uses the start and end coordinates of the selected source, but you can change these values as desired. When you change these values, the system automatically updates the dimensions:

- *Start Coordinate*: Type the starting coordinate for the subsource (the top-left corner of the subsource) in the format "x,y", where x is the horizontal coordinate and y is the vertical coordinate. Note that the coordinate system uses the top-left corner as the origin (0,0). Also note that when video is shown at 100-300%, a tooltip appears under the cursor, indicating its position.
- *End Coordinate*: Type the ending coordinate for the subsource (the bottom-right corner of the subsource) in the format "x,y", where x is the horizontal coordinate and y is the vertical coordinate. Note that the coordinate system uses the top-left corner as the origin (0,0).

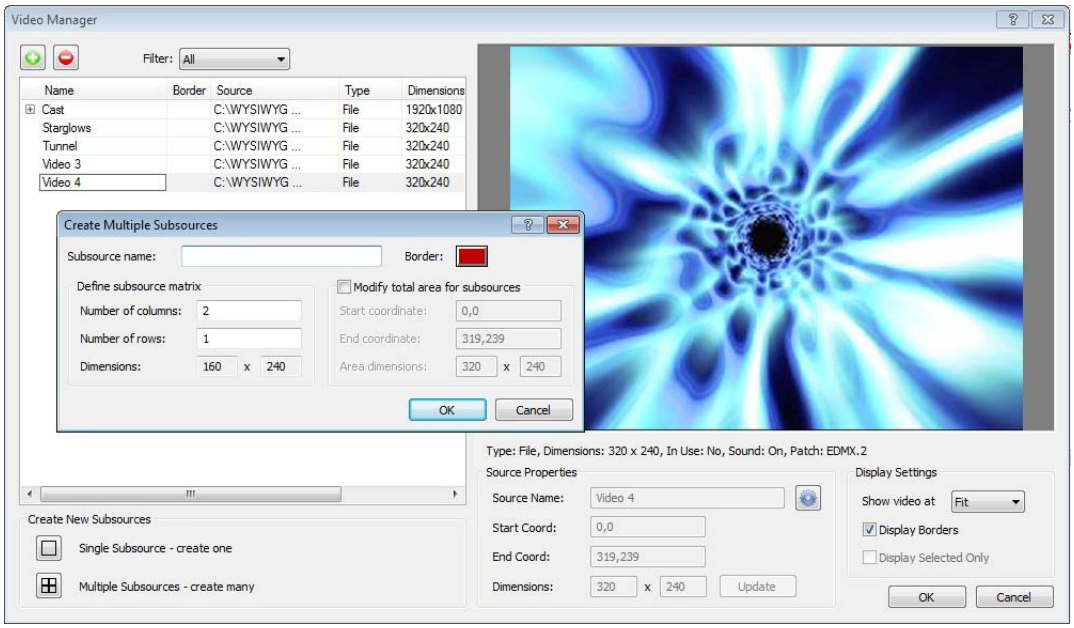
5 Click the Border box to specify the color of the border that will appear around the subsource when displayed.

6 Click *OK*.

To create multiple video subsources

1 In the left pane of the Video Manager, click to highlight the video for which you want to create subsources.

2 In the bottom left corner of the window, click *Multiple Subsources - create many*.



3 Type the subsource name.

4 Define the subsource matrix by specifying the number of columns and rows. The dimensions of the subsources are displayed for your reference:

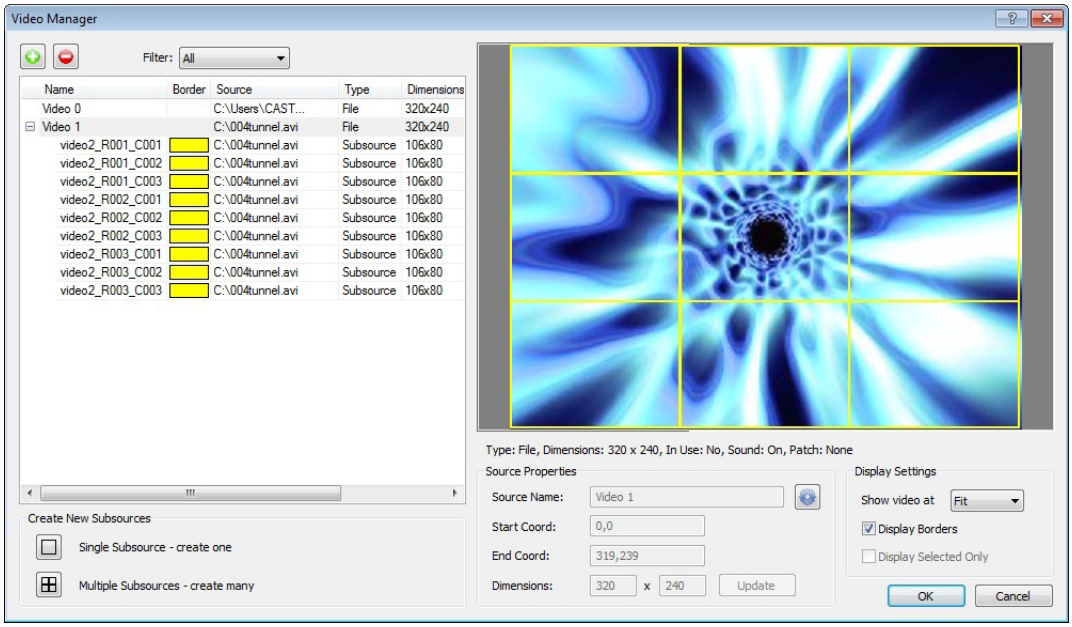
- *Number of columns*: Type the number of columns of subsources to be generated. If the video resolution cannot be equally divided by this number, the remaining pixels will be left at the end (for the horizontal pixels this means the ones at the far right, and for the vertical pixels this means the ones at the bottom).
- *Number of rows*: Type the number of rows of subsources to be generated. If the video resolution cannot be equally divided by this number, there will be a few gaps in the generated subsource grid.

5 By default, the subsources are created by taking the overall source area (height & width) and evenly dividing it by matrix provided. However, you can modify the total area by changing the start/end coordinates and, therefore, the overall dimensions used to create the subsources. To do so, click the check box beside *Modify total area for subsources*, and then enter the appropriate values:

- *Start Coordinate*: Type the starting coordinate for the subsource (the top-left corner of the subsource) in the format "x,y", where x is the horizontal coordinate and y is the vertical coordinate. Note that the coordinate system uses the top-left corner as the origin (0,0). Also note that when video is shown at 100-300%, a tooltip appears under the cursor, indicating its position.
- *End Coordinate*: Type the ending coordinate for the subsource (the bottom-right corner of the subsource) in the format "x,y", where x is the horizontal coordinate and y is the vertical coordinate. Note that the coordinate system uses the top-left corner as the origin (0,0).

6 Click *OK*.

Result: The following graphic shows a video with six subsources (three columns and three rows).



Notes:

- Subsources are always listed below the source from which they were created.
- The table in the Video Manager provides a convenient way of displaying source/subsource information; all sources and their subsources are grouped, and can be expanded/collapsed.
- You can rearrange the columns in the table by clicking the column header and dragging it to the new position.
- You can adjust the width of columns by clicking the column border and dragging the column to its new width.

Changing the properties of video sources and subsources

Note: If you change the source after the subsources were created, and the new source has different dimensions that do not match the subsources dimensions, the affected subsources are highlighted in yellow (and a dialog appears to notify you as such).

When you select a row in the Video Manager table, the properties of the video source or subsource are displayed in the Source Properties area of the window. You can change various properties of the video source and subsources as follows:

- To change the border color of video subsources, in the table, click the appropriate border color and then click on the drop-down box that appears to select the new color. Click *Update*.
- To change the name of the video source or subsource, in the table, click the appropriate row. Type the new name, and then click *Update*.
- If you have already applied a video to multiple objects, and then you want to replace the source file with something different, you can make the change once in the Video Manager and have it automatically applied to all related objects (i.e., you don't have to individually apply the changes to each object's properties). To do so, you change the underlying source file to the Video name.
- You can define a subsource either by its start/end coordinates or by its dimensions:

- To resize the subsource, click the appropriate row in the table. Change the start/end coordinates as desired, and then click *Update*. Note that if you change the coordinates, the system updates the dimensions automatically.
- To change the dimensions of a subsource, click the appropriate row and then type the new dimensions. Note that when you change the dimensions, the system will automatically add to the start coordinate to determine a new end coordinate. Click *Update*.

Note: You can use the Quick Video/Image Tool to apply the Video Subsources. For details, see “Using the Quick Video/Image Tool” on page 348.

To configure a new video source for streaming video

To create a new video source for streaming video, you use the Video Manager. There are a couple of ways of opening the Video Manager (for example, from the New Screen window, or from the **Tools** menu); the following procedure lists one possible way.

Note: You can only incorporate live video if you have WYSIWYG Perform; you cannot use a live video from a video capture device or video streaming via CITP/MSEX if you have WYSIWYG Design.

1 Click **Tools** > **Video Manager**.

2 Click the *New* icon in the upper-left corner.

Result: The Video Source window opens.

3 In the *Name* box, type a descriptive name for the video.

- To play a video from a file, click the *Video File* option button, and then click *Browse* to locate the file.
- To capture a live video stream from an external source, such as a web cam or a video capture device that is attached to your computer, click *Video Capture*, and then use the drop-down arrow to select the appropriate device. (This option is available only if you are running WYSIWYG Perform.) WYSIWYG will detect what standard resolutions the capture device can support, and display them in the *Resolution* drop-down box; click this drop-down to choose your preferred resolution.
- If you select a video capture option, configure the Input and Resolution values for the device:
  - *Input*: Choose between various inputs found on video capture devices (e.g., HDMI, DVI, S-Video, and so on).
  - *Resolution*: Select the resolution of the incoming video, provided that multiple resolutions are available from the selected input.

Note: The video capture device must already be configured on your computer and must be using WDM drivers. Also, it must not be currently in use by another application.

- To stream video from a media server on your network, first ensure that the *CITP Interface* option is enabled on the Application Options > Additional Interfaces tab, as detailed on page 393. (This option is only available if you are running WYSIWYG Perform.) Then, click *CITP Video Stream* and click *Browse* to select one of the media servers from the network. If you have just enabled this option, you will need to restart WYSIWYG before the CITP interface is turned on.

Note: In order for the media servers to be detected, they must be in the same logical network as your WYSIWYG computer and any and all firewalls (on your WYSIWYG computer, on the media servers, if applicable, and so on) must be disabled. For more information about how to correctly assign IP addresses for this purpose, consult your Media Server and/or Microsoft Windows documentation.

4 If your media server is capable of outputting several video streams at once, you may select which output/stream to connect to by entering the number of the output/stream in the *Video Num* box. If you leave the default value of *0*, WYSIWYG will connect to the first output; if you enter the number *1*, WYSIWYG connects to the second output/stream of the same media server; the number *2* connects to the third output/stream, and so on.

5 To control video playback via DMX, click *DMX Patch*, and then use the drop-down arrow to select the appropriate patch universe to control the video. In the box to the right of the drop-down list, type the starting DMX address.

Note: A patch universe cannot be created from within the Video Manager dialog; if you wish to patch video control to DMX, you must create the necessary patch universe(s) before you open the Video Manager.

6 To mute the sound of the video, in the Video Manager table, click the checkbox in the *Mute* column. If you do not select this checkbox, the video's sound will play.

7 Click *OK*. If the video is available, it connects automatically.

8 Click *OK* to save your changes and close the Video Manager.

Image Manager

Introduction

You can store images as sources in WYSIWYG using the Image Manager. By assigning textures to saved sources instead of to specific images, you can change or modify the texture of multiple objects at the same time using the Image Wizard.

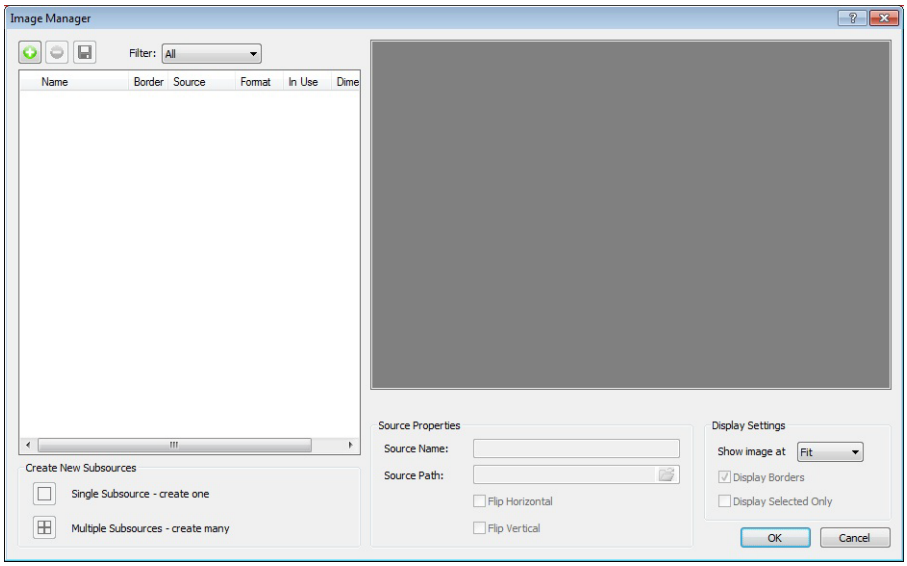
The Image Manager also enables you to perform minor image modifications, such as flip horizontal or flip vertical to images without permanently changing the source file. This feature is useful in situations where the texture is flipped as a result of incorrect surface winding. For details, see “Drawing surfaces” on page 153.

You can use the Image Manager to create subsources of an image (i.e., divide the image up), which can then be applied to objects such as surfaces, screens, LED walls.

To access the Image Manager

1 In CAD mode, select **Tools** > **Image Manager**.

Result: The Image Manager appears.

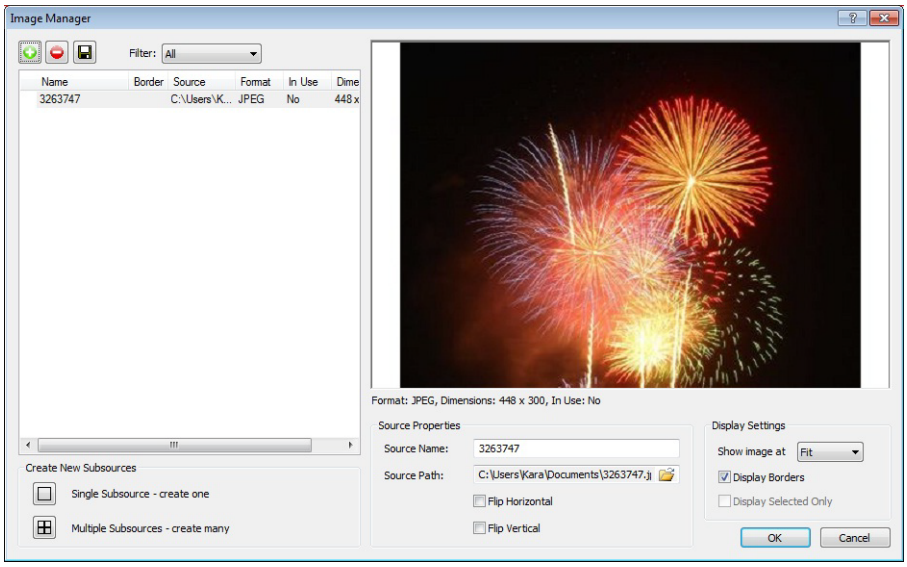


To create a new image source in the image manager

1 In the Image Manager, click *New*.

2 Browse to locate the image on your computer, and then click *OK*.

Result: The Image is added as a source.



Notes:

- *Source Name*: The image's file name.
- *Source Path*: The path to the physical file. Click the file open icon to select a new file.
- *Flip Horizontal*: Click this option to flip the image source horizontally. For details, see "To flip an image source" on page 344. Note that you cannot flip image subsources.
- *Flip Vertical*: Click this option to flip the image source vertically. For details, see "To flip an image source" on page 344. Note that you cannot flip image subsources.
- *Dimensions*: The file's dimensions in pixels.
- *In Use*: Informs you of whether the image source is in use. If the image source is currently in use, it cannot be deleted from the Image Manager.
- Display Settings
  - *Show image at*: Select the zoom level for the image. You can choose to "fit" the image in the display window, or show it at a percentage of its actual size: 100%, 200% or 300%.
  - *Display Borders*: Select this option to show the subsource borders in the display.
  - *Display Selected Only*: When you have selected a subsource in the table, click this check box to *only* show the subsource in the display area; if you do not select this checkbox when a subsource is selected, the entire source is shown (with the subsource borders highlighted in the display if you selected *Display Borders*).

To flip an image source

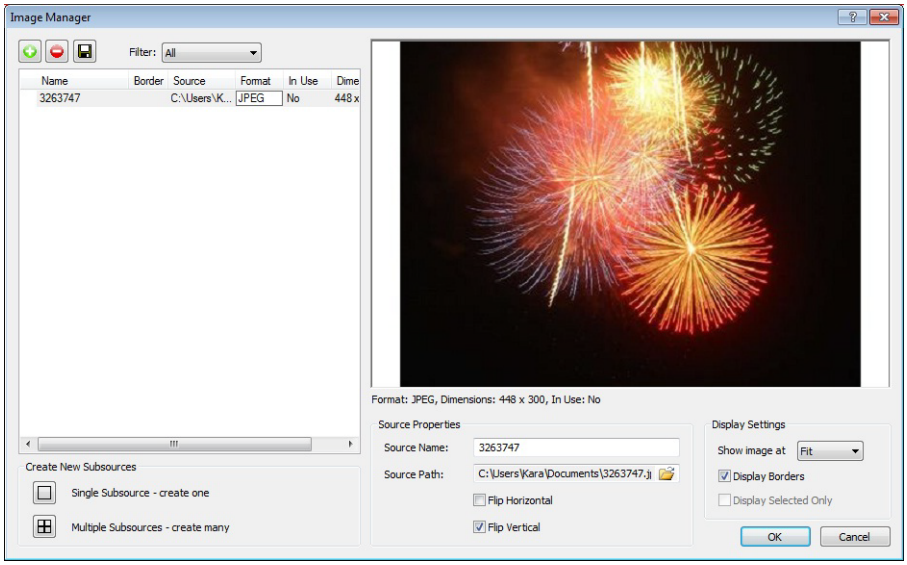
In situations where a surface was drawn in a clockwise direction (see "Drawing surfaces" on page 153), the texture may appear to be "flipped" either horizontally or vertically. To avoid this problem, flip the image source using the *Flip Horizontal* or *Flip Vertical* checkboxes, as shown below. Note that you cannot flip image subsources; only the actual image source can be flipped.

1 Open the Image Manager.

2 In the table to the left, select the image source that you want to flip.

3 Based on the direction in which you want to flip the image, select either *Flip Horizontal* or *Flip Vertical*.

Result: The image is flipped, as shown in the following sample graphic, which was flipped vertically.



To export an image source

After flipping the image, you may want to save your changes to a new image file as follows:

1 Open the Image Manager.

2 In the table to the left, select the image source that you want to export.

3 Click the Save icon in the top left of the window.

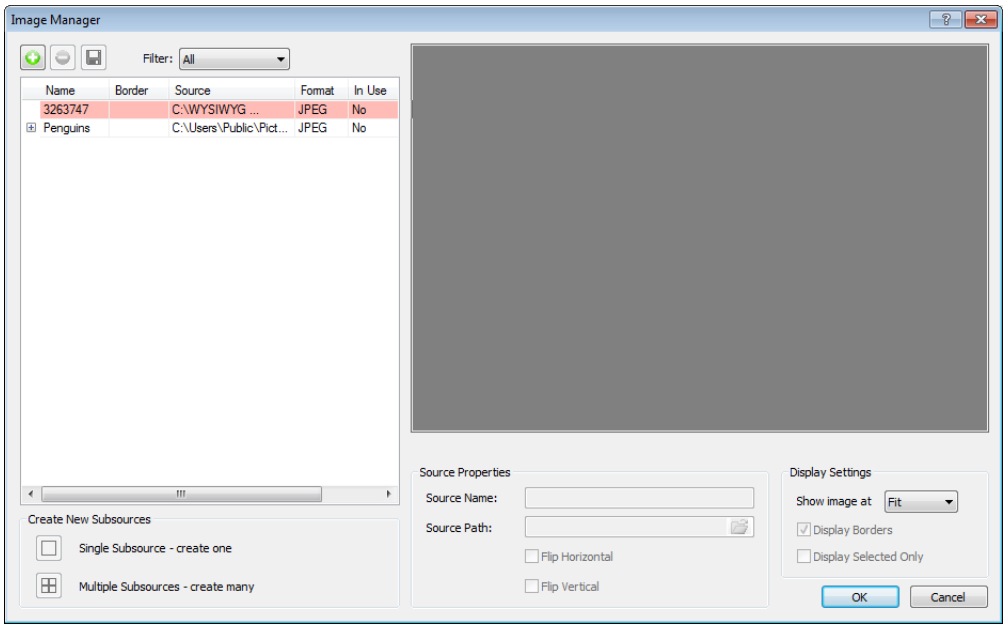
4 Type a name and select a file type for the new image, or accept the default file type of PNG.

Note: When exporting an image source the default file type is PNG, but you can choose a different file type, if desired; when exporting a subsource, you cannot choose the file type. The subsource is exported to the same file type as its source.

5 Click *OK*.

Missing Images

If you have turned off the option *Save external textures within the document* in Application Options > File Options, when you move image files from their original location, they may not load from within WYSIWYG. When this happens, the image source will be highlighted in red, as shown in the following graphic:



To address this issue, select the image source highlighted in red, click the folder icon beside the Path box, and browse to select a new image. When you are finished, click *OK*.

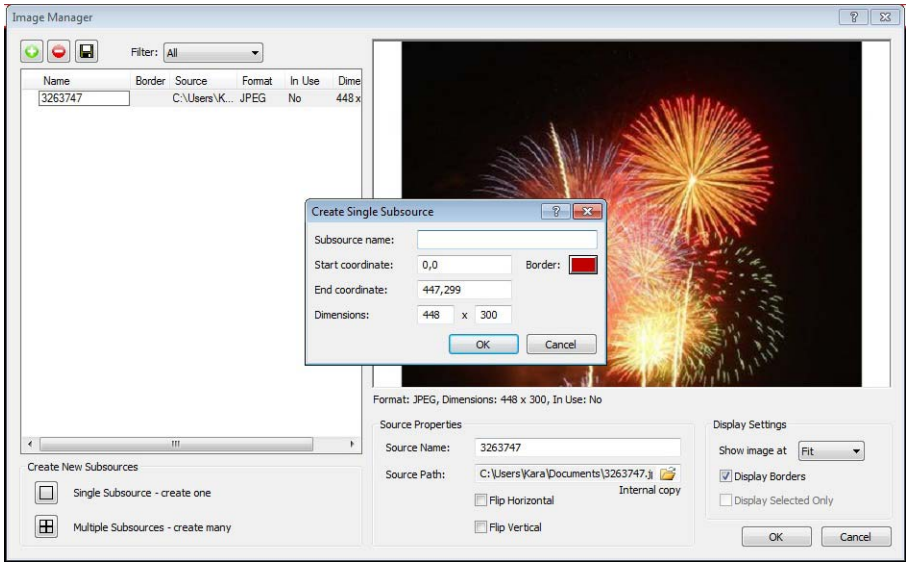
Creating Image Subsources

Just as with videos, you can create one subsource or multiple subsources from an image using the following procedures. Image subsources are useful in scenarios where you need to apply part of an image to one object and another. Normally, to do so you would need to open the image in a photo editing software program, make your edits, crop the image, and then import two images. However, with the image subsource creation feature, you can just import the source file, and create the necessary subsources within WYSIWYG. For example, you could use this procedure if you have a photo that you want to break into quadrants (2 x 2), and then apply each quadrant to a separate surface, screen or LED wall, etc.

To create a single image subsource

1 In the left pane of the Image Manager, click to highlight the image for which you want to create a subsource.

2 In the bottom left corner of the window, click *Single Subsource - create one*.



3 Type the subsource name.

4 By default, the system uses the start and end coordinates of the selected source, but you can change these values as desired. When you change these values, the system automatically updates the dimensions.

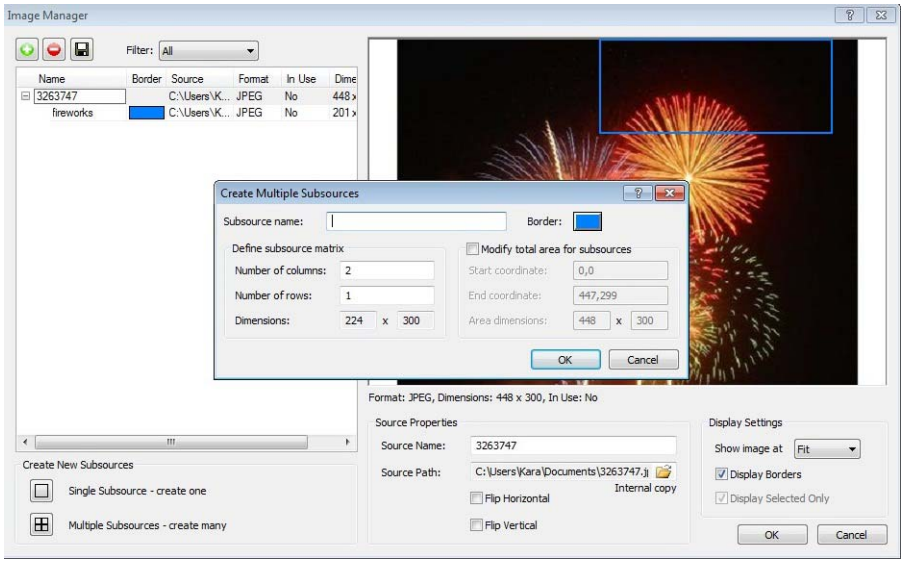
5 Click the Border box to specify the color of the border that will appear around the subsource when displayed.

6 Click *OK*.

To create multiple image subsources

1 In the left pane of the Image Manager, click to highlight the image for which you want to create subsources.

2 In the bottom left corner of the window, click *Multiple Subsources - create many*.



3 Type the subsource name.

4 Define the subsource matrix by specifying the number of columns and rows. The dimensions of the subsources are displayed for your reference.

5 By default, the subsources are created by taking the overall source area (height & width) and evenly dividing it by matrix provided. However, you can modify the total area by changing the start/end coordinates and, therefore, the overall dimensions used to create the subsources. To do so, click the check box beside *Modify total area for subsources*, and then type the new coordinates.

Notes:

- Subsources are always listed below the source from which they were created.
- The table in the Image Manager provides a convenient way of displaying source/subsource information; all sources and their subsources are grouped, and can be expanded/collapsed.
- You can rearrange the columns in the table by clicking the column header and dragging it to the new position.
- You can adjust the width of columns by clicking the column border and dragging the column to its new width.

Changing the properties of image sources and subsources

Note: If you change the source after the subsources were created, and the new source has different dimensions that do not match the subsources dimensions, the affected subsources are highlighted in yellow (and a dialog appears to notify you as such).

When you select a row in the Image Manager table, the properties of the image source or subsource are displayed in the Source Properties area of the window. You can change various properties of the image source and subsources as follows:

- To change the border color of image subsources, in the table, click the appropriate border color and then click on the drop-down box that appears to select the new color. Click *Update*.
- To change the name of the image source or subsource, in the table, click the appropriate row. Type the new name, and then click *Update*.
- If you have already applied an image to multiple objects, and then you want to replace the source file with something different, you can make the change once in the Image Manager and have it automatically applied to all related objects (i.e., you don't have to individually apply the changes to each object's properties). To do so, you change the underlying source file to the Image name.
- You can define a subsource either by its start/end coordinates or by its dimensions:
  - To resize the subsource, click the appropriate row in the table. Change the start/end coordinates as desired, and then click *Update*. Note that if you change the coordinates, the system updates the dimensions automatically.
  - To change the dimensions of a subsource, click the appropriate row and then type the new dimensions. Note that when you change the dimensions, the system will automatically add to the start coordinate to determine a new end coordinate. Click *Update*.

  Note: You can use the Quick Video/Image Tool to apply the Image Subsources. For details, see "Using the Quick Video/Image Tool" on page 348.

Using the Quick Video/Image Tool

The Quick Video/Image Tool enables you to quickly configure which video sources / subsources or images to attach to selected objects like screens and LED Walls in your drawing.

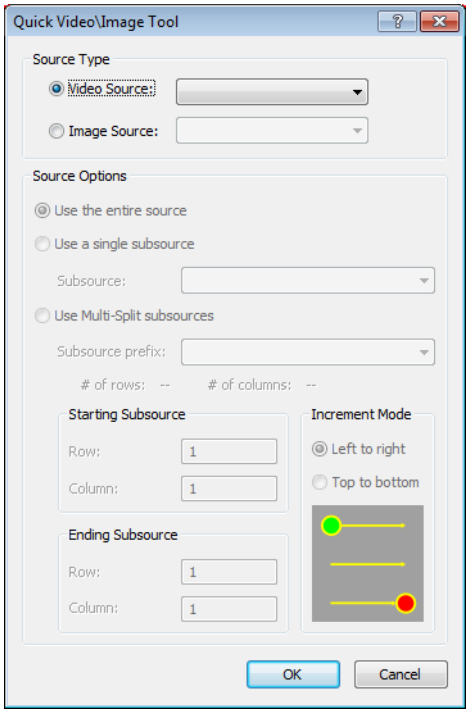
To use the Quick Video/Image Tool

Note: Before you use the Quick Video/Image Tool, you must already have created the video source and/or subsources or images that you want to apply to the selected object(s). For details, see “To configure a new video source for streaming video” on page 341 and “To create a single video subsource” on page 338.

1 From the **Tools** menu, choose **Quick Video/Image Tool**.

Tip: Alternately, you can click the *Quick Video/Image Tool* icon on the Tools toolbar.





2 In the Source Type section, select whether you want to apply either a video or image source, and then choose the source from the appropriate drop-down box.

3 Under Source Options, choose from one of the following options:

- *Use the entire source*: Select this option if you wish to apply the entire source (as opposed to its subsources). Proceed directly to step 12.
- *Use a single subsource*: Select this option if you wish to apply a single subsource, and then choose the subsource from the drop-down box. Proceed directly to step 12.
- *Use Multi-Split subsources*: Select this option if you wish to apply Multi-Split subsources. This option will start at a specified subsource and increment the rows and/or columns until it reaches an ending subsource, at which point the tool will finish automatically.

4 If you have chosen *Use Multi-Split subsources* above, from the *Subsource prefix* drop-down box, select the Multi-Split subsource prefix to apply.

5 Note the # of rows and columns shown. These values denote the number of rows and columns available for this Multi-Split subsource prefix. The Starting and Ending Row, and the Starting and Ending Columns may not exceed these numbers.

6 In the *Starting Subsource - Row* box, enter the row of the starting subsource. It may not exceed the number of rows for this subsource prefix.

7 In the *Starting Subsource - Column* box, enter the column of the starting subsource. It may not exceed the number of columns for this subsource prefix.

8 In the *Ending Subsource - Row* box, enter the row of the ending subsource. It may not exceed the number of rows for this subsource prefix, and cannot be less than the starting subsource's row value.

9 In the *Ending Subsource - Column* box, enter the column of the ending subsource. It may not exceed the number of rows for this subsource prefix, and cannot be less than the starting subsource's column value.

10 Under Increment Mode, choose the appropriate option:

- Click *Left to right* if you want subsources to be applied from left to right. This way, subsources on the same row will be applied first, and the current row will move down once all subsources on the current row have been applied.
- Click *Top to bottom* if you want subsources to be applied from top to bottom. This way, subsources on the same column will be applied first, and the current column will move right once all subsources on the current column have been applied.

11 Click *OK*. The cursor will show a “Q”, indicating that Quick Tools have been activated.

12 Click an object, or multiple objects in sequence to apply the video/image source or subsource(s) that you have chosen.

The Flight Case

Introduction

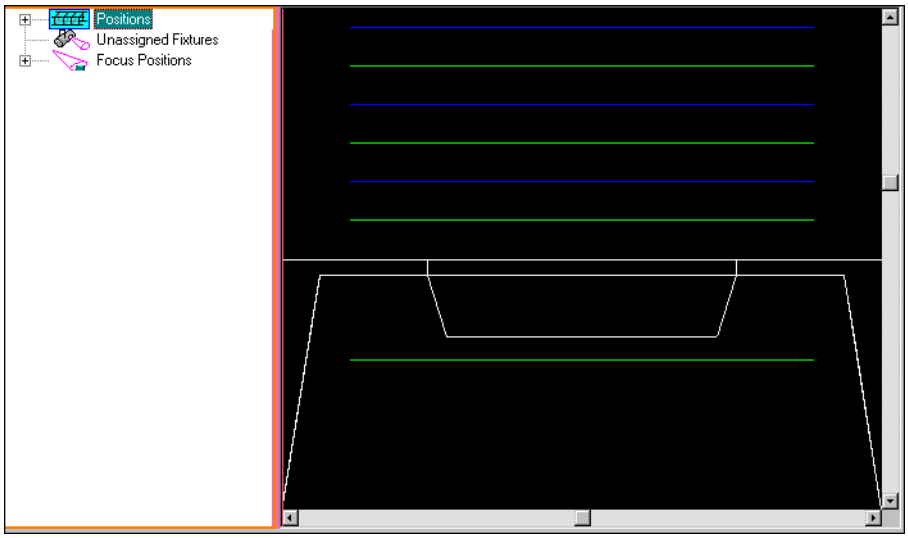
The Flight Case is a special view used to hold unhung fixtures, hang structures, and focus positions created or inserted in Data mode but not yet placed on the drawing. This is an ideal way to work from a set inventory as you can add all of your fixtures in Data mode, and then click and drag the fixtures out of the Flight Case until you have exhausted your stock.

To open the Flight Case

1 From the **Tools** menu, choose **Flight Case**.

Note: You can open the Flight Case only in a wireframe view.

Tip: The Flight Case layout is preconfigured with the flight case and a wireframe view.



How objects get into the Flight Case

As you enter fixtures in the Data mode spreadsheets, they appear in the Flight Case under the *Unassigned Fixtures* branch. When you type position names in the Position Manager, the individual positions appear under the *Position* branch. As you assign fixtures to those positions, they will move from the *Unassigned* branch to under the proper position. All fixtures that appear in the Flight Case are unhung fixtures, which means they are not in your drawing (regardless of whether they have been assigned a position or not).

If you create focus positions in the Data mode by typing labels in the focus box of the spreadsheet, those focus positions will appear under the *Focus Positions* branch of the Flight Case. Unhung fixtures assigned to those focus positions will appear under their respective Flight Case headings.

To learn more about inserting objects in Data mode, please refer to Chapter 6, “Data mode.”

Entering and modifying objects in the Flight Case

- You can insert fixtures and focus positions directly into the Flight Case using the Library, *Draw* menu, or appropriate tools on the *Draw* toolbar.
- You can change a fixture's assigned position by selecting the fixture in the Flight Case and dragging it to another position within the Flight Case.
- You can change a fixture's assigned focus position by selecting the fixture and dragging it between Focus Positions within the Flight Case.

Transferring objects from the Flight Case to the drawing

The items that appear in the Flight Case can be transferred to your drawing by clicking and dragging.

To hang fixtures from the Flight Case

1 Open the Flight Case.

2 Click and hold on the desired fixture.

3 Drag the fixture into the wireframe view and place it on a hang structure.

Note: If a fixture appears in the flight case under a position branch, that fixture must be hung on that position. Fixtures with no position or unassigned fixtures may be hung on any hang structure.

4 Release the mouse button.

5 Repeat for the remaining fixtures.

Tip: You can hang all the fixtures on one position by dragging the entire position out of the Flight Case.

Selecting a fixture in your drawing and pressing the Backspace key will return the fixture to the Flight Case. The fixture will retain the position assignment.

To insert a hang structure from the Flight Case

If you drag a position from the Flight Case, you will create a pipe.

1 Expand the *Positions* branch to see all the hang structures.

2 Click and hold on the position that you want to insert.

3 Drag the pipe onto the drawing.

4 Release the mouse button.

5 Use the pop up dialog box to answer the questions about the trim height and length of the pipe, or use the *Interactive* button to draw the pipe in interactive mode.

Result: If you do not choose interactive mode, the pipe is inserted centered on the origin at the trim height specified.

Note: Multiple positions can have the same name.

For more information about inserting pipes please refer to "Drawing pipes" on page 162.

To insert a focus position from the Flight Case

1 Expand the *Focus Positions* branch in the flight case to see all the Focus positions that have not yet been placed in the drawing.

2 Click and hold on the focus position that you want to insert into the drawing.

3 Drag the Focus position onto the drawing.

4 Release the mouse button to place the focus position.

Flight Case properties

The Flight Case properties affect which fixture attributes are shown.

To modify the Flight Case properties

1 From the **Options** menu, choose **View Options**.

Tip: You can also use the *View Options* tool on the *Standard* toolbar. 

2 Click the *Show Details* tab.

3 Select the fixture attributes that you want displayed in the Flight Case.

Note: The *Unit Number* is displayed by default, however, you can select all the attributes if you want.

Rendering

Introduction

At any time in CAD mode, you can generate a rendering of your set. Note that renderings in CAD mode are primarily intended for rendering your set with environmental lighting, but without theatrical lighting. However, if there are any fixtures selected on your plot, they are displayed in your rendering.

Note: To queue your render jobs and process them independently, you can use the Background Rendering Manager (all levels of WYSIWYG except Report and Perform - Console Edition). For details on this feature, see “Background Rendering Manager” on page 508.

To render a lighting look

1 In the working area of the screen, display the lighting look that you want to render.

2 From the **Design** menu, choose **Render**.

Tip: You can also click the *Render Wizard* tool in the *Design* toolbar. 

Result: The Render Wizard opens and guides you through a series of options. Click *Next* and *Back* to navigate through the Render Wizard.

3 Click *Finish*.

Result: WYSIWYG generates the rendering of your lighting look or cue. If the rendering is saved internally, it is available as an image in Presentation mode.

Note: For more information, see “Rendering” on page 497.

Chapter 6

Data mode



Data mode enables you to view and edit the data for all your fixtures. Data mode operates like many other spreadsheet programs, so many of the concepts will be familiar to those who have used a spreadsheet program.

In this chapter

Layout tabs	357
Spreadsheets	358
Patch	366
Errors	376



Layout tabs

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The Data mode contains the following layouts:

- Spreadsheet - The work area contains a spreadsheet of data currently in the show database.
- H Select - The work area is split horizontally, with a wireframe view above and a spreadsheet view below.
- V Select - The work area is split vertically, with a wireframe view on the left and a spreadsheet view on the right.
- Patch - The work area is split, with a wireframe view above and the patch window below.
- Errors - The work area contains a spreadsheet displaying fixtures with errors.

Note: The wireframe views in Data mode are used for fixture selection only. Any selections made in the spreadsheet are visible on the CAD drawing and vice versa. However, to make any drawing modifications, you must return to CAD mode.

Spreadsheets

Introduction

The Data mode displays your fixture information in spreadsheet format. WYSIWYG compiles many spreadsheets that are accessible from the columns shortcut bar. All of these spreadsheets contain the same information, but are sorted and filtered differently. Each column in the spreadsheet represents one fixture attribute. The following information is compiled.

Data fields

Note: Data fields identified with an asterisk (*) are non-editable fields.

- **Spot** - This is an assigned identifier number usually used for automated fixtures. A spot number is required for automated fixtures when using AutoFocus.
- **Channel** - This is the assigned control channel number you will use at your control console to control the fixture. For moving lights, the channel number recorded in WYSIWYG is the starting channel number.

Note: Please note that to use the AutoFocus feature properly, for most consoles, the Channel number that appears here must match the starting number of the fixture's DMX address. For example, if your Mac 500 is patched to universe "A" at address 101 (i.e., the Patch field reads "A.101"), you must enter the number 101 into this fixture's Channel field. This must be done manually because the automatic sequential numerical data entry method does not apply in this case. For details, see “To input sequential numerical data” on page 360.

- **Position** - This is the hanging position for the fixture. Positions must be entered in the Position Manager.
- **Unit** - The unit number identifies the fixture’s location on its respective hanging position.
- **Type** - This is the fixture name.
- **Lens** - This is the lens type.
- ***Hookup** - This identifies the component of a multi-circuit fixture or other device, such as a scroller (for example, intensity, control).
- **Purpose** - The purpose is a custom note that is most commonly used to describe how this fixture is being used in your show. For example, “SL Side”, “Diagonal Backs”. Purpose is an attribute of the fixture. It is not possible to assign multiple purposes for multi-circuit fixtures.
- **Color** - This is the assigned gel color number or scroller identification.
- **Gobo** - This is the assigned gobo number.
- **Focus** - This is the fixture’s focus position.
- **Patch** - This is the fixture’s assigned DMX channel number. This field is mandatory for simulation activity in Live mode. One show can have multiple DMX universes. A patch entry must be notated *universe.#*, where *universe* is a letter, number, or other label identifying the universe or output and # is the DMX channel number. For example, “A.1” or “Dim.26”.
- **Circuit Name** - This is an identifier note for the circuit box or multi-cable.
- **Circuit Number** - This is the assigned circuit or multi-cable tail number.
- **Dimmer** - This is the assigned dimmer number.

- ***Offset** - This field identifies the fixture's location on the hanging structure. It is a distance measurement referencing the pipe's end or center point or another point as specified.
- **Pan** - A focus attribute measured in degrees, defining the positioning of the fixtures yoke.
- **Tilt** - A focus attribute measured in degrees, defining the positioning of the fixture within the yoke.
- **Spin** - A focus attribute measured in degrees, defining the fixture's yoke positioning in relation to the hang structure where 0 is down-hung, 90 is side-hung, and 180 is over-hung for example.
- ***Weight** - This is the fixture's weight. A fixture's weight can only be modified through the Library Browser.
- **Lamp Type** - This is the lamp type.
- **Notes** - This is a custom notes field.
- ***Footnotes** - This feature is currently disabled.
- ***# of Data Channels** - This is the total number of DMX channels required by the fixture.
- ***# of Color Frames** - This is the number of color frame slots that the fixture has.
- ***# of Lamps** - This is the number of lamps required by the fixture.
- ***Circuit Type** - This describes what type of device the unit should be plugged into, for example, regular dimmer, scroller power supply.
- ***Wattage** - This is the wattage in watts of the lamp.
- ***Model** - This is the fixture type.
- ***Cost** - This is the fixture's cost or rental cost. This field is used to estimate a show budget. A fixture's cost can only be modified through the Library Browser. For more information on setting costs, refer to page 297.
- ***Status** - This is the fixture's status relative to your drawing. If the fixture is HUNG it is in your drawing. If a fixture is UNHUNG, it is in the flight case. In fixture count reports, all fixtures are counted regardless of their status (unless a filter is applied).
- ***Console** - This identifies which console is controlling the fixture. This field references the binding settings in the device manager in Live mode.
- **Layer** - This field indicates which layer the fixture is drawn on.
- ***Tag** - This is an internal code used for importing and exporting data to/from third party programs.
- ***Owner** - This feature is currently disabled.
- Manufacturer - This field indicates the manufacturer of the fixture.
- **X** - This field indicates the X coordinate of the fixture's position
- **Y** - This field indicates the Y coordinate of the fixture's position
- **Z** - This field indicates the Z coordinate of the fixture's position

Working in the spreadsheet

Data may be entered in a number of ways within the WYSIWYG file. The plot can be created, and then edited, or the data may be entered in a spreadsheet, and then placed on the plot. Any entries or changes are reciprocated throughout the file; changes made in Data mode will be updated in CAD mode and visa versa.

To add or modify data

Type the desired value in the appropriate cell.

Tip: You can add or modify chunks of information at the same time by selecting a series of cells and typing.

If you are entering sequential numerical values for a field such as patch, you can use incremental data entry to facilitate your work. WYSIWYG will calculate the next available value based on the number of required channels for the previous fixture.

To input sequential numerical data

1 Type the first value in the appropriate cell.

2 Press and hold ALT and press the down-arrow for each incremental entry you want to make.

To choose a new value

1 Select the appropriate cell.

2 Right-click and choose **Pick value from the list**.

Result: This action opens a dialog with a drop-down menu listing all of the values currently used in your file for you to choose from.

3 To make a new selection, click *Library* to open the appropriate library.

Note: This feature is only available in certain fields.

Tip: This feature is especially useful for fields such as color, gobo, or other parameters that reference library items for which you do not know the exact value.

Find and replace tool

The find and replace tools are used for locating and modifying data, and are column specific. The tools only search for data in the column where the tool was activated and not throughout the entire spreadsheet.

To use the find tool

1 Select a cell in the column in which you want to use the *Find* tool.

2 From the **Edit** menu, choose **Find**.

Tip: You can also use the *Find* tool on the *Data* toolbar. 

3 Type your search criteria.

- Select the *Match whole word only* check box to search for data matching exactly what you have typed as opposed to data containing parts or all of what you have typed.
- Select the *Match case* check box to enable a case sensitive search where uppercase and lowercase letters will affect the results.

4 Click *Find Next*.

Result: Cells that match your search criteria are highlighted with the first match active.

To use the replace tool

1 Select a cell in the column in which you want to use the *Find* and *Replace* tool.

2 From the **Edit** menu, choose **Replace**.

Tip: You can also use the *Replace* tool on the *Data* toolbar. 

3 Type your search criteria. Enter a value to find and a value to replace it with.

- Select the *Match whole word only* check box to search for data matching exactly what you have typed as opposed to data containing parts or all of what you have typed.
- Select the *Match case* check box to enable a case sensitive search where uppercase and lowercase letters will affect the results.

4 Click *Find Next*, *Replace*, or *Replace All*.

Result:

- If you click *Find Next*, cells that match your search criteria are highlighted with the first match active.
- If you click *Replace*, cells that match your search criteria are highlighted and the first match is replaced with the value specified in step 3. You can continue clicking *Replace* to replace each match in turn.
- If you click *Replace All*, all matches are replaced with the value specified in step 3 and no cells are highlighted.

Inserting fixtures in Data mode

Any fixtures inserted in Data mode are assigned the status “unhung” and are placed in the Flight Case. The Flight Case allows you to drag and drop “unhung” fixtures onto your drawing. For more information about the Flight Case, refer to “The Flight Case” on page 351.

To insert fixtures in Data mode

1 Click the *Fixture* tool  on the *Data* toolbar.

2 Navigate to the desired fixture.

3 In the *Multiple* box, type the number of fixtures of that type required.

4 Click *Insert*.

Result: The fixtures are inserted below the last entry in the spreadsheet.

Tip: If you have a shortcut created for the desired fixture, you can right-click on the shortcut and choose **Insert** or **Insert Multiple**.

Inserting focus positions in Data mode

If you type a new value into the focus field of a fixture, you are essentially creating a focus position. That focus position is now available in the Flight Case for you to drag and drop into your drawing.

For more information on the Flight Case, refer to “The Flight Case” on page 351.

Inserting positions in Data mode

If you type a new value into the position field of a fixture, the *Pick a value from the list* dialog box is automatically displayed. This is because a position cannot exist in WYSIWYG unless it is recorded in the *Position Manager.* You can select from the list of positions that already exist or you can click *Manager* to open the *Position Manager* to create a new position.

If you are making this change to a fixture that was previously hung on a different position, that fixture will by unhung and sent to the *Flight Case* under its new position field. From there, you can drag it back onto the drawing.

If the position does not yet exist in the drawing, you must draw a hang structure and assign it the appropriate position name before you will be able to hang the fixture again. For more information on drawing hang structures, refer to “Hang structures” on page 257

For more information on drawing items from the Flight Case, refer to “Entering and modifying objects in the Flight Case” on page 352.

Inserting color in Data mode

To insert color in Data mode

1. Click in the color field of the fixture for which you want to assign color.
2. Right-click and choose **Pick value from the list**.
3. Use the pull-down menu to select a color among those you have already used in your plot, or click *Library* to open the color library.

   Tip: You can bypass steps 2-3 and type in the value if you already know the color that you want to assign (for example, R54, L112, and so on)

   WYSIWYG accommodates multiple color entries for one fixture as follows:

   - “Color1 & Color2” yields two color frames with one gel in each.
   - “Color1 + Color2” yields one color frame with two gels in it.
   - “Color1 / Color2” yields one color frame with one split gel in it.

Data filters

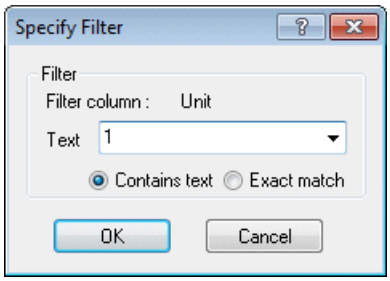
Data filters are used for locating or isolating selective blocks of information. When a data filter is applied to a spreadsheet, only the units that met the filter criteria are displayed. Filters are applied to columns of data.

To apply a data filter

1 In the spreadsheet, right-click on the column heading you want to filter.

2 Choose **Add Data Filter**.

Result: A dialog box is displayed, showing the column name and the available filters.



3 Select the value that you want to filter. You can also type in a value and choose whether you want to filter that specific text, or filter for entries that contain that text.

Result: The spreadsheet refreshes, displaying only fixtures that meet the filter criteria.

To remove data filters

1 Right-click on a column heading.

2 Choose **Remove Data Filter**.

Result: The spreadsheet returns to its unfiltered state.

Customizing spreadsheets

There are different ways to sort and view your data. First, consider the different spreadsheets available on the *Columns* shortcut bar.

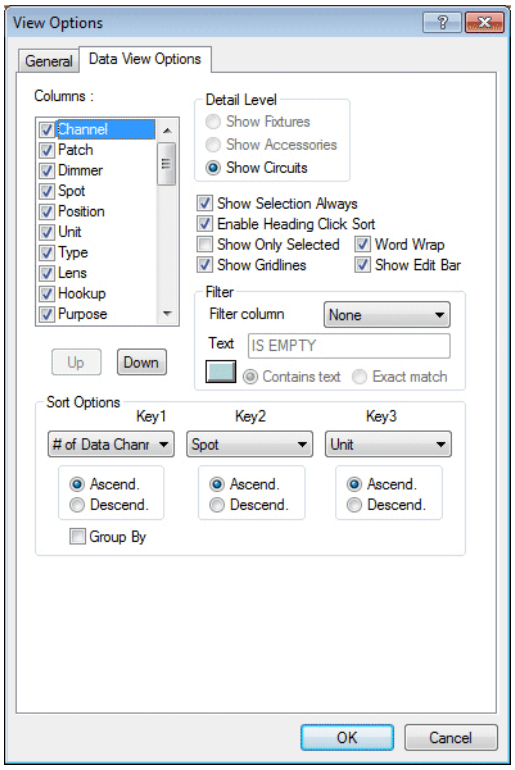
Alternatively you can modify a spreadsheet to suit your needs. Customizing a view allows you to change how the data is displayed and sorted.

To modify a data sheet

1 From the **Options** menu, choose **View Options**.

2 You can also click the *View Options* tool on the *Standard* toolbar. 

3 Click the *Data View Options* tab.



4 To adjust a column's location in the spreadsheet, highlight the appropriate column heading in the *Columns* box, and then click the *Up* or *Down* button to relocate it in the data view.

5 To hide a column in your view, deselect the check box beside it.

6 To specify how entries should be sorted, choose the desired column headings in the *Key 1*, K*ey 2*, and *Key 3* drop-down lists. When fixtures have the same value in the first sort key, the spreadsheet is then sorted by the values of the second sort key.

7 Click *Ascend* or *Descend* to sort the criteria in ascending or descending order, respectively.

8 Select the *Show Selection Always* check box to ensure that any fixtures selected in a plot view will automatically be selected in a data view.

9 Select the *Enable Heading Click Sort* check box to enable the ability to sort the spreadsheet by clicking on the column headings while working in the spreadsheet.

10 The *Show Only Selected* option will filter the spreadsheet to display only the fixtures that are selected in the Wireframe view.

11 Select the *Show Gridlines* check box to display the columns in tabular format, with lines separating the data.

12 Select the *Word Wrap check box to* view all the text on a line in the spreadsheet.

13 Select the *Show Edit Bar* check box to display the edit bar at the top of the spreadsheet.

14 The *Filter* section has the same functionality as the data filters discussed in "Data filters" on page 362.

15 Select the *Group By* check box to divide the spreadsheet into sections/pages, one section/page for each value in the Key 1 field.

Creating a new spreadsheet

To create a new sheet

1. In the shortcut bar, click the *Columns* tab.
2. Right-click in the shortcut bar and choose **New Spreadsheet.**
3. Type a name for the new spreadsheet, and then click *OK*.
4. Scroll to the bottom of the list of *Column* shortcuts. Your new spreadsheet name should be at the bottom of the list. Click on the shortcut to view your spreadsheet.

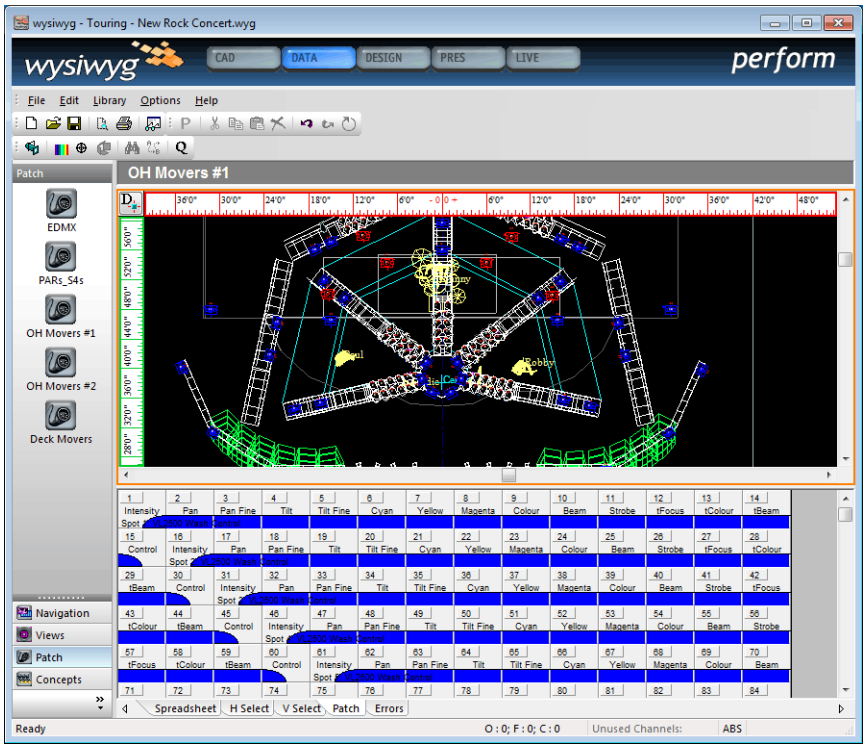
   Note: It may be easier to clone an existing spreadsheet and modify it than to start a new one from scratch. See “To clone a shortcut” on page 57 for more details.

Patch

Introduction

The Patch view is a graphical representation of the patch field. It is used for two purposes:

- to create and/or edit a patch for the lights, video, or moving scenery in your drawing
- to monitor incoming DMX levels when using WYSIWYG Perform for simulation



Working in the patch view

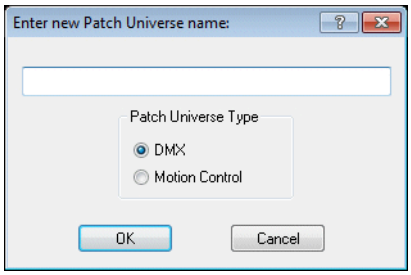
You can view one patch universe at a time. As you add new patch universes in either a spreadsheet view or a CAD drawing (using Quick Tools), their shortcuts are created automatically. You can also create new patch universes directly from the shortcut bar.

To create a new patch universe

1 Click the *Patch* layout tab.

2 Right-click in the patch shortcut area and choose **New Patch**.

Result: A dialog box is displayed, prompting for the type of universe that you want to create, and the name of the new universe.



3 Type the name of the patch universe. Patch universes can be named anything. For example, “A”, “Dimmers”, “Dim”, “Movers”, “Scrollers”, and so on.

4 Select the type of patch universe based on the type of console or system that you are using, either a DMX console, or a motion control system.

5 Click *OK*.

Result: The patch shortcut appears in the Patch tab.

Notes:

- To change, delete, or modify a patch universe, right-click the shortcut icon for the universe that you want to change. Click on the command that you want to perform.
- To display the contents of a patch universe, click on the appropriate shortcut icon. The name of the universe is displayed at the top left corner of the Wireframe view.

Fixture patching methods

There are multiple ways to patch fixtures in WYSIWYG. The following section will guide you through some of the ways fixtures can be patched.

To patch fixtures in the patch view

1 Click and hold the desired fixture in the Wireframe view.

2 Drag the fixture to the desired patch location.

3 For re-patching, repeat the above steps, or click and drag the fixture’s patch information to the new location.

To unpatch fixtures in the patch view

1 Select the fixture by clicking on the first channel (start cell) for that fixture. It should become highlighted.

2 Right-click and choose **Unpatch**.

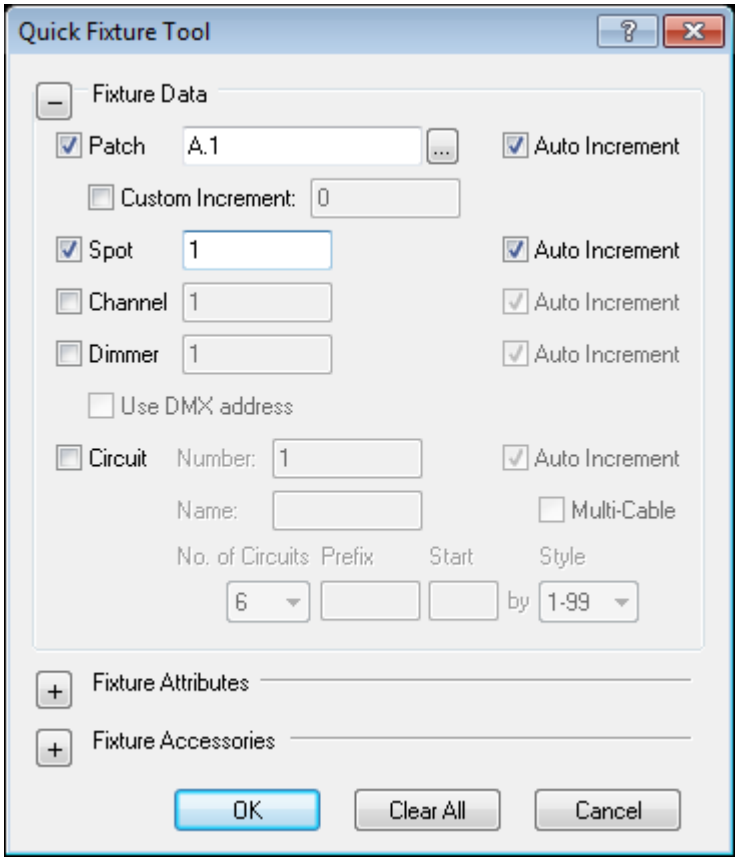
Result: The fixture is unpatched; the value in the patch field for that fixture is deleted.

To patch fixtures and assign spot IDs using Quick Tools

Using Quick Tools, fixtures can be patched and assigned spot IDs by clicking on their **Wireframe** view image. The first fixture clicked will be assigned the patch and ID as configured in Quick Tools. The next fixture clicked will be assigned the next available channel in the patch and the next fixture ID in the sequence until you are finished using Quick Tools.

1 In **CAD** mode, in **Wireframe** view, go to **Tools > Quick Tools > Quick Tools**. Alternately click the Quick Tools icon in the Tools toolbar.

Result: The *Quick Fixture Tool* window will open.



2 In the *Quick Fixture Tool* window, in the **Fixture Attributes** section, mark the **Spot** check box.

3 In the **Spot** field, enter a number which will be the first fixture ID used in the sequence.

4 In the **Fixture Data** section, mark the **Patch** check box.

5 In the **Patch** field, enter the universe and channel the first fixture will patching to, separated by a period.

Example: “A.1” would patch to universe A, chapter 1.

6 Click *Ok*

Result: When a fixture is clicked on, it will be assigned a Universe and channel to operate on. The first fixture clicked will be assigned the to the channel number entered in Quick Tools. Subsequent fixtures clicked will be assigned the next sequential channel and Spot ID number.

7 Click on a fixture to patch it to a universe and channel. Clicking on each fixture that you want patched and assigned a spot ID.

8 To stop patching fixtures, right click.

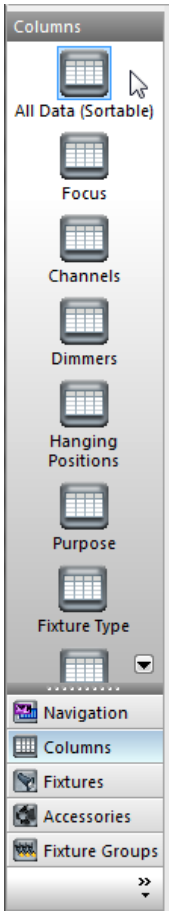
9 In the menu that appears, click **Finish Quick Tools**.

To patch fixtures and assign spot IDs using the data spreadsheet

In **DATA** mode, the **Spreadsheet** view has information on all fixtures created in the WYSIWYG project. Using the spreadsheet you can edit the patch information and Spot ID directly in the table.

1 In **Data** mode, click on the **Spreadsheet** tab.

2 In the **Columns** section on the left side of the window, click *All Data (Sortable)*.



Result: All fixture data will be displayed in the spreadsheet.

3 In the **Patch** column, enter the appropriate patch information for each fixture.

4 In the **Spot** column, enter a unique Spot ID for each fixture.

5 It is possible to assign sequential values to fixtures quickly by entering a value for the first fixture at the top of the list, then entering **ALT + Down Arrow**. The fixture below will be assigned the next available value. This can be repeated as necessary. If this method is used to assign values, it is recommend that fixtures are first sorted so fixtures are organized. Fixtures can be sorted in order of position, or Unit # along a pipe/truss.

To patch the control of a video source

In WYSIWYG, you patch the *control* of a video source, not the actual video. This means that you can use the console device to control the progress of a live video stream or a video file by making the video play, pause, or rewind; you cannot use WYSIWYG to change different aspects of the video itself.

To control the video source, the DMX patch is allocated three channels. For details on controlling the video with the console device in Live mode, see “To control a DMX patched video source with a console” on page 531.

1 In the shortcut area, click the *Patch* tab.

2 Click the patch shortcut to which you have linked control of the video source.

Result: The video patch appears in the patch universe.

3 Open the console device that you have connected to the video DMX patch.

4 Using the console controls, set the appropriate levels for the three channels. For details, see “To control a DMX patched video source with a console” on page 531.

5 Save your settings.

6 In the Shaded tab of Live mode, you can now use the console device to control the progress of the video. For details, see “To control a DMX patched video source with a console” on page 531.

To patch a motion axis to a DMX console device

In WYSIWYG, you can patch a motion axis to which one or more objects has been attached, and then use a console device or a motion control system to control the progress of the objects along the axis.

You have the choice of patching the axis to either a DMX patch universe or a motion control universe, based on the type of console or system you will be using to move the objects. For details on patching the axis to a motion control universe, see “To patch a motion axis to a motion control system” on page 371.

To control the object’s movement with a DMX console device, the DMX patch is allocated two channels, one each for coarse and fine movement. The DMX value represents the object’s percentage of movement. For example, if it is at 0, then the object is at 0% of its range of motion; if it is at 100, then the object has moved 100% of its range of motion. For details on controlling the object with the console device in Live mode, see “To control a DMX patched motion axis with a console” on page 532.

Note: Before performing the following procedure, the motion axis must already be linked to the patch universe. For details, see “To patch a motion axis to a DMX universe” on page 172.

1 In the shortcut area, click the *Patch* tab.

2 Click the patch shortcut to which you have linked control of the moving object.

Result: The moving scenery patch appears in the patch universe.

3 Open the console device that you have connected to the DMX patch (for example, the Mini Console).

4 Using the console controls, move the slider of the first or second channel to advance the object to the desired position on the motion axis.

Note: The first channel is for coarse movement and can be used to advance the object quickly to any position along the entire path, from start to finish. The second channel is for fine movement. Its movement range is very slight and depends on the position that you have set on the first channel. For details, see “To control a DMX patched motion axis with a console” on page 532.

5 In the Shaded tab of Live mode, you can now use the console device to control the object’s movement along the axis. For details, see “To control a DMX patched motion axis with a console” on page 532.

To patch a motion axis to a motion control system

In WYSIWYG, you can patch a motion axis to which one or more objects has been attached, and then use a console device or a motion control system to control the progress of the objects along the axis.

You have the choice of patching the axis to either a DMX patch universe or a motion control universe, based on the type of console or system you will be using to move the objects. For details on patching the axis to a DMX patch universe, see “To patch a motion axis to a DMX console device” on page 370.

When controlling the object’s movement with a motion control system, the patch indicates the ID on the motion control system that corresponds to the WYSIWYG axis ID, the axis name, and the position of the object on its path. When connected to a system (or to WYSIWYG’s Motion Control Console), you can watch the object’s position change in patch view as it moves along its path.

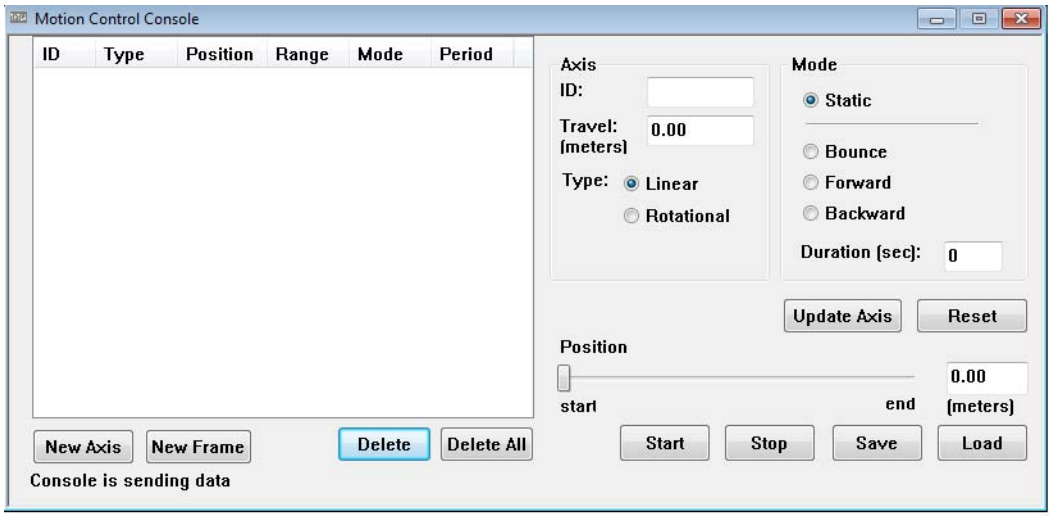
Note: Before performing this procedure, the motion axis must already be linked to the patch universe. For details, see “To patch a motion axis to a motion universe” on page 172.

1 In the shortcut area, click the *Patch* tab.

2 Click the patch shortcut to which you have linked control of the moving object.

Result: The motion control patch appears in the patch universe.

3 Open the console device or motion control system that you have connected to the motion control patch (for example, the Motion Control Console, shown below).



4 If you are using WYSIWYG’S Motion Control Console, in the *ID* box, type the motion control ID that you selected when configuring the axis. If you are patching a Frame, enter the name of the frame.

Note: If you do not know the ID, right-click on the axis, and then click **Properties.** Click the Axis tab. The ID appears in the *Motion Patch - ID* box.

5 In the *Travel* box, type the distance (in meters for linear axes, in degrees for rotational axes) that you want the object to travel along or around the axis. This can be the full length/angle of the axis, or only a portion of it if you do not want the object to travel along/around the full path. For linear axes, if you type a value that is longer than the actual axis, the object stops at the end of the physical axis.

6 Click the *Linear* or *Rotational* option button according to the type of motion axis that you are patching.

7 In the Mode area, select the type of movement for the object:

- **Static -** Select this option if you want to specify the precise location of a static object on the motion axis. You can then use the position slider at the bottom of the window to adjust the position and control the movement of the object manually.
- **Bounce** - Select this option if you want the object to move forward and backward in a continuous loop along the motion axis.
- **Forward -** Select this option if you want the object to move forward along the axis from start to finish, and then start over again at the beginning in a continuous forward loop.
- **Backward -** Select this option if you want the object to move backwards along the axis from finish to start, and then begin over again in a continuous backwards loop.

8 If you have chosen any *moving* Mode value (that is, any value *except* Static), you can specify the length of time (in seconds) over which you want the full range of motion to take place. Type this value in the *Duration* box. The larger the number you type in this box, the slower the object moves.

9 Select *New Axis* if you are adding a linear or rotational frame. Select *New Frame* if you are adding a frame to the patch.

10 Click *Save* to save your settings.

Note: If you make any changes to the axis settings and save the changes, you can revert to the previous settings by clicking *Load*.

11 To see the object moving along the axis, you must first ensure that the Motion Control Console or the motion control system that you are using is connected and that the patched axis is bound. For details, see “To connect to a console” on page 526 and “To control a motion-control patched motion axis with a motion control system” on page 532.

Reading the patch

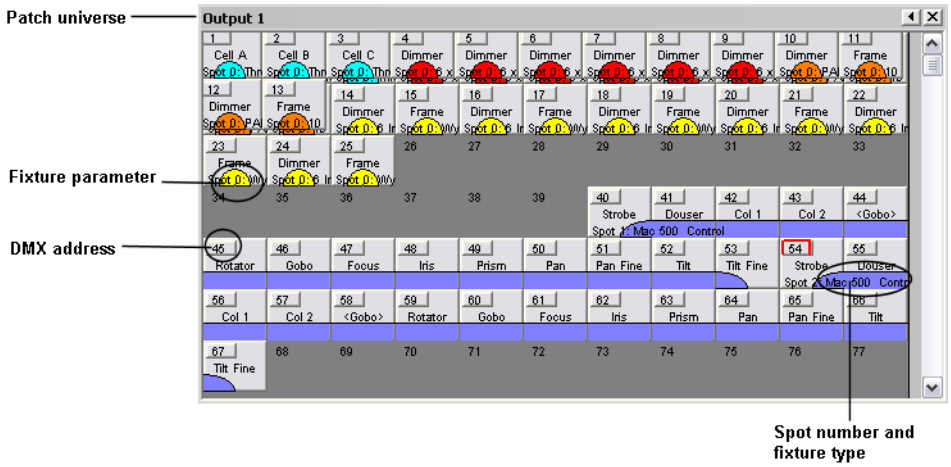
Each box in the patch universe represents one DMX channel. Fixtures are represented by colored bumps or bars that span the number of required DMX channels. The color of the bar is the same as the fixture’s CAD layer.

The number in the top left corner is the DMX address.

The spot number and fixture type are identified across the bottom of the box(es).

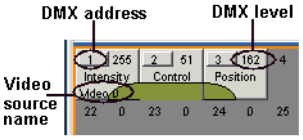
The fixture parameter controlled by each channel is identified in the center of the box.

If you are connected to a console and are receiving DMX signals, the incoming DMX levels are displayed in the top right corner of each box.



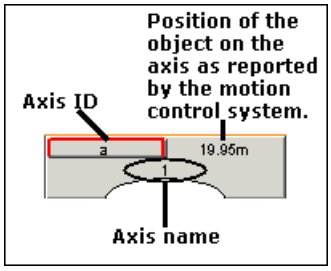
Reading the patch for streaming video sources

If you have patched a live video source or file, the patch appears as shown below:



Reading the patch for moving scenery

If you have patched a motion axis, the patch appears as shown below:



- For linear motion axes, the position of the object is reported in either metric or imperial units (for example, 16m).
- For rotational axes, the value is shown in degrees (for example, 42°).
- If the axis is unrecognized by the motion controller, then the object position is shown as a question mark (?).

Note: If the axis type is mismatched between that which you have drawn and the type that you choose in the Motion Control Console (for example, if the actual axis is linear, but you choose rotational in the Motion Control Console), then the cell in patch view turns to red.

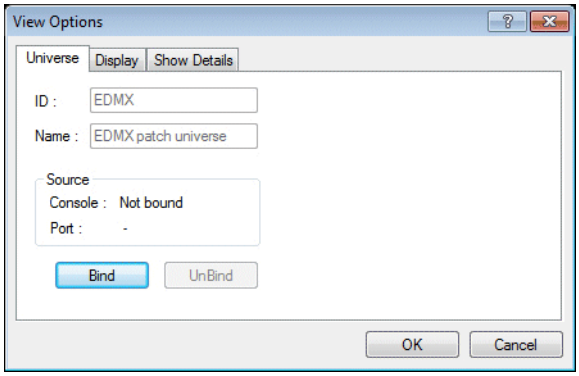
Patch view properties

You can view and modify patch universe properties such as the name, display details, and tooltip information that is displayed in a patch universe.

To view the properties of a patch universe

1 Right-click on the patch icon, and then select **View Options**.

Result: The *View Options* dialog box is displayed.



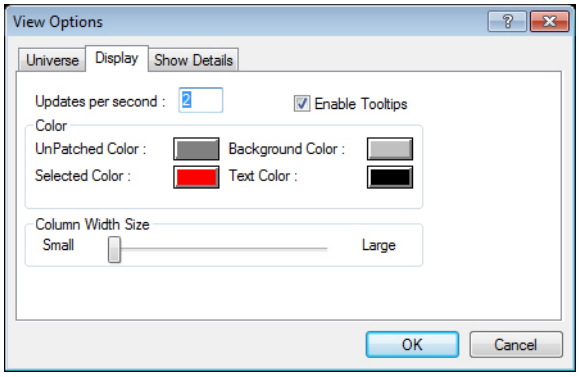
Universe tab

Options on the *Universe* tab affect the binding/unbinding of the patch universe to a specific console or device output.

- *ID*: The name of the patch universe shortcut.
- *Name*: The full name of the patch universe or port ID.
- *Source*: The lighting control console or device and port to which the patch universe is bound.
- *Bind*: Associates the patch universe to a console or device output. This applies only when a console or device has been inserted in the Device Manager.
- *Unbind*: Removes the association of the patch universe with the specified source.
  Tip: You can open the *Device Manager* without having to change to Live mode by double-clicking on the *AF* shortcut on the status bar.

Display tab

Options on the *Display* tab set properties such as the color of a patch universe.



- *Updates per second:* The frequency of updates from the console to the patch universe.
- *Enable Tooltips*: Select this check box to display information about the patch and the fixture when you hover over the start cell.
- *UnPatched Color:* The color of a cell in the patch universe that is not patched.
- *Background Color:* The color of the background of the patch view.
- *Text Color*: The color of the text that is displayed in the patch universe.
- *Selected Color*: The highlighted color of a selected fixture in the patch universe.

- *Column Width Size:* Adjust the slider to resize the patch universe cells to the desired width.

To change the width of the patch cells

You can adjust the width of the cells in DMX and Motion Control patch universes.

1 Right-click anywhere in the patch universe and choose **View Options**.

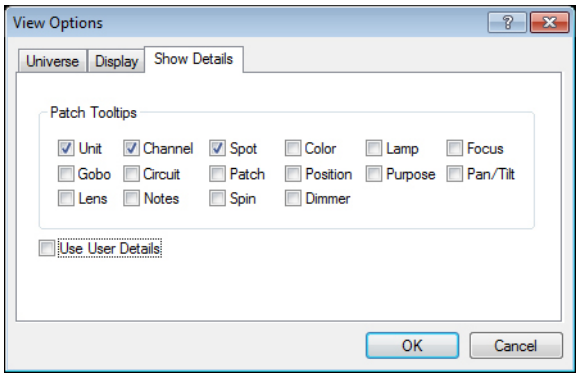
2 Click the Display tab.

3 Adjust the Column Width Slider to the preferred size.

4 Click *OK*.

Show Details tab

Options on the *Show Details* tab define the information to be shown in the tooltips when tooltips are enabled.



- *Use User Details:* Select this check box to reference the settings in *User Options* (under the *Options* menu). Click to clear the check box and specify the tooltips that you want displayed.



Errors

Introduction

WYSIWYG sports an “on the fly” error checking engine. This means that as you are putting your lighting design together, WYSIWYG is continually checking for data errors such as multiple dimmers in one circuit, multiple channels in one dimmer, and various missing information.

At any time, you can view the error reports that are available on the *Errors* shortcut bar. Error reports that have errors displayed will have an exclamation mark (!) after their name. You can edit/fix these errors on the spot; editing data on the error reports works the same way as the data spreadsheets explained in the previous section. As soon as you fix an error in an error spreadsheet, the entries are disabled to show that it has been fixed.

An indicator on the status bar signals you when your document contains errors. This feature can be turned on or off. WYSIWYG is also set up to trigger an error notification when errors are detected. Error notifications can be turned on or off.

Errors

WYSIWYG checks for the following errors:

Error	Definition
Dimmers > Circuit	Fixtures assigned to the same circuit have been assigned different dimmer numbers.
Patches > Circuit	Fixtures assigned to the same circuit have been assigned different patch addresses.
Channels > Circuit	Fixtures assigned to the same circuit have been assigned different channel numbers.
Patches > Dimmer	Fixtures assigned to the same dimmer have been assigned different patch addresses.
Channels > Dimmer	Fixtures assigned to the same dimmer have been assigned different channel numbers.
Channels > Patch	Fixtures assigned to the same patch address have been assigned different channel numbers.
Spots > Patch	Fixtures assigned to the same patch address have been assigned different spot numbers.
Missing Position	The position field is empty.
Missing Unit	The unit field is empty.
Missing Circuit Name	The circuit name field is empty.
Missing Circuit #	The circuit number field is empty.

Error	Definition
Missing Dimmer	The dimmer field is empty.
Missing Patch	The patch field is empty.
Missing Channel	The channel field is empty.
Missing Spot	The spot field is empty.
Missing Color	The color field is empty.
Missing Gobo	The gobo field is empty.

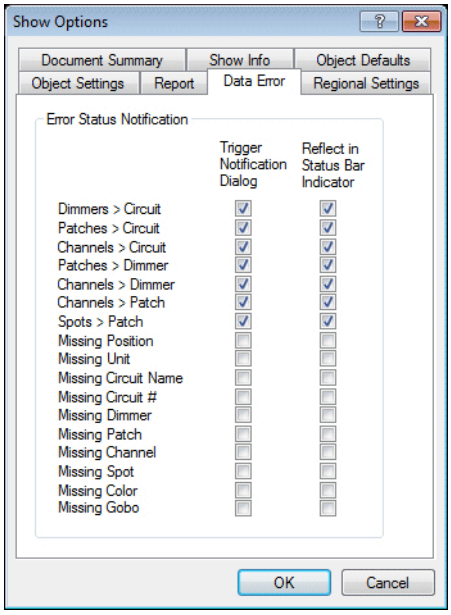
Error notifications

Error notifications occur when WYSIWYG detects an error for which an error notification trigger has been set.

To set error triggers

1 From the **Options** menu, choose **Show Options**.

2 Click the *Data Error* tab.

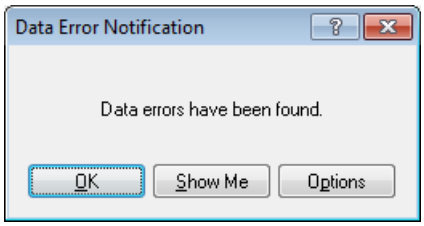


3 Select the errors for which you want to trigger an error notification and/or signal.

4 Click *OK*.

To manage error triggers

1 When an error notification is triggered, the following dialog box appears:



2 From this dialog box, you can perform one of the following actions:

a. Acknowledge the error, and then click *OK* to close the notification.

b. Acknowledge the error, and then click *Show Me* to open a new window that automatically navigates you to the appropriate error report.

c. Acknowledge the error, and then click *Options* to reset the error notification triggers as described above.

Error spreadsheet properties

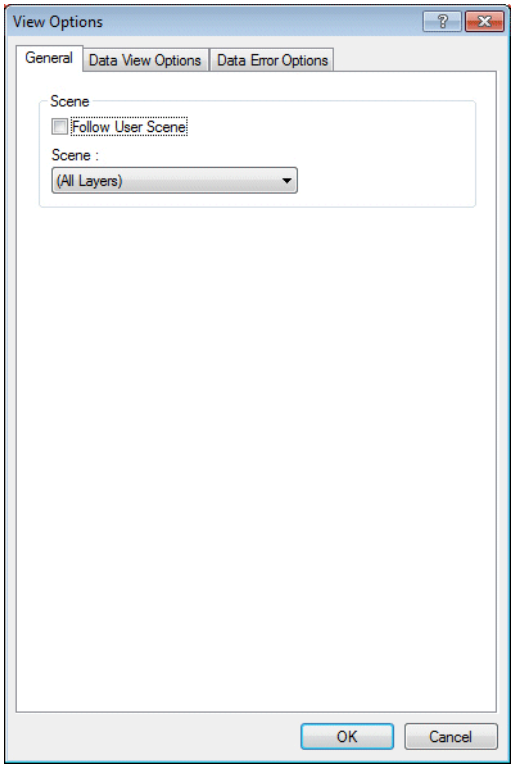
Error spreadsheets may be sorted and modified in a similar manner as other spreadsheet views.

To modify an error sheet

1 Click the *Errors* layout tab.

2 From the **Options** menu, choose **View Options**.

Result: The *View Options* dialog box is displayed.



For information on the *General* and *Data View Options* tab, refer to “Customizing spreadsheets” on page 363.

On the *Data Error Options* tab, set triggers for the error that is displayed in the *Name* box on the *General* tab. For more information on triggers, refer to “To set error triggers” on page 377.

Chapter 7

Managing your WYSIWYG document



This chapter provides information on how to work with your WYSIWYG document.

In this chapter

File menu 381

Options menu 389

WYSIWYG Viewer 410

File menu

Introduction

The **File** menu contains the basic commands required to manipulate your WYSIWYG document.

The **File** menu is the same in all the modes.

New command

The **New** command creates a new show document. If another show document is currently open, you will be prompted to save changes to that document before a new untitled file is created. Only one show document may be open at a time.

To create a new document

From the **File** menu, choose **New**.

Tip: You can also use the *New* tool on the *Standard* toolbar. 

Open command

The **Open** command opens a previously created show document. If another show document is currently open, you will be prompted to save changes to that document before another show document is opened. Only one show document may be open at a time.

To open an existing document

1 From the **File** menu, choose **Open**.

2 Select the file, and then click *Open*.

You can open the following file types:

- WYSIWYG files (.wyg)
- WYSIWYG template (.wyt)
- Vivien files (.vvn)
- Vivien template files (.vvt)
- WYSIWYG backup files (.bak)
- Wysiwygv2 files (.wys)
- DWG files (.dwg)
- DXF files (.dxf)
- LW2 files (.lw2)
- SketchUp files (.skp)

BTX Export

This command exports the current show to a BlackTrax file (.btx) which can be opened by BlackTrax System.

DWG/DXF Export

This command exports the current show document to a DWG or DXF formatted file type. DWG and DXF formats are used to transfer documents to AutoCAD or other compatible drafting applications.

To export to DWG or DXF format

For details, see “To export to DWG/DXF in 2D” on page 125.

Save command

The **Save** command saves the open document to the same file name and location (folder, disk, and so on) under which it was previously saved. If you are saving the document for the first time, this command will perform the Save As function.

To save your document

From the **File** menu, choose **Save**.

Tip: You can also use the *Save* tool on the *Standard* toolbar. 

Save As command

The **Save As** command is used to save the current document to a new file name, a new file type, or new destination, such as a floppy disk.

To save the current document

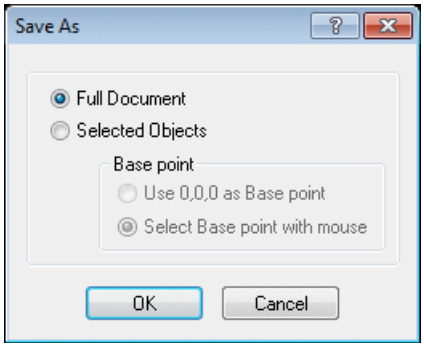
1. From the **File** menu, choose **Save As**.
2. In the dialog box that opens, browse to the location where you want to save the file. Ensure that the destination appears in the *Save in* box.
3. In the *File name* box, type a name for the file.
4. In the *Save as type* box, choose the saved file type. There are a number of choices available.
5. Click *Save*.

   Note: WYSIWYG allows you to save an object or group of objects contained within your document as a separate file.

To save an object

1. Ensure the object(s) to be saved are selected.
2. From the **File** menu, choose **Save As**.

   Result: The following dialog box is displayed.



3 Click *Full Document* to save the current document to a new file name, file type, or new destination. You are prompted to enter the destination and file name of the file.

4 Click *Selected Objects* to save the selected object(s) as a separate WYSIWYG (.wyg) document.

5 Click *Use 0,0,0 as Basepoint* to align the incoming drawing at 0,0,0 with the existing drawing.

6 Click *Select Basepoint with mouse* to use the mouse to select the basepoint for the merge of the CAD drawings.

7 Click *OK*.

To save a WYSIWYG document as a Vivien file

1 From the File menu, choose Save As.

2 In the dialog box that opens, browse to the location where you want to save the file. Ensure that the destination appears in the *Save in* box.

3 In the *File name* box, enter a name for the file.

4 In the *Save as type* box, choose Vivien Files (*.vvn).

5 Click *Save*.

Template files

The WYSIWYG **Save As** command lists a file type called.WYT. WYSIWYG template files are similar to the type of template file you may use on your word processor to set up fax cover sheets, memos, or letters. Once you have drawn your venue with all of its lighting positions and everything else that is common to all shows, save it as a .WYT file in the \Templates directory. Each time you start a new drawing, that file will be available as a template to start from.

You can also enter all your lighting equipment inventory into the *Flight Case* and save that as a template. When users start their drawings with your template, they will not have to choose from an endless list of fixtures but rather just what you have in stock.

This is also the ideal option for saving your preferred working settings. Template files save all the shortcuts including custom reports, plots, and so on. However, toolbar settings are not saved in template files since they are saved by the application.

A list of most recently opened template files appears under the heading *Create New Plot* on the WYSIWYG welcome screen. The *more...* link will prompt you to open a template file from the WYSIWYG Templates directory.

To save a template file

1 In an active view, and with no objects selected, from the **File** menu, choose **Save As**.

2 In the dialog box that opens, browse to the location where you want to save the file. Ensure that the destination appears in the *Save in* box. To ensure that your template file is available on the WYSIWYG Welcome screen, save your file to the WYSIWYG/Templates directory.

3 In the *File name* box, type a name for the file.

4 In the *Save as type* box, choose **WYSIWYG Template files (*.wyt)**.

5 Click *Save*.

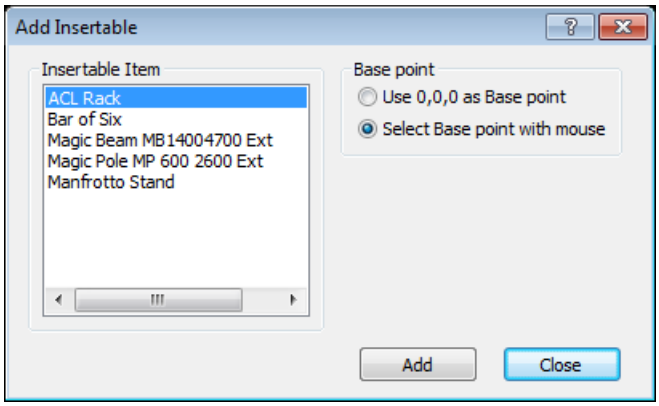
Add insertable

WYSIWYG ships with several files that contain sets of pre-configured objects. These can be merged in quickly from this dialog.

To use the Add Insertable command

1 Click **File** > **Add Insertable**.

Result: The Add Insertable window appears.



2 In the left pane, click the item that you want to insert.

3 Click the appropriate base point option.

4 Click OK.

5 If you chose to select the base point with your mouse, click in your drawing to select the base point and insert the item.

Purge command

Purge enables you to lower your file size by ridding the file of unused data such as old images, old renders, unused Layers, lighting positions, imported items from AutoCAD or Sketchup, and so on.

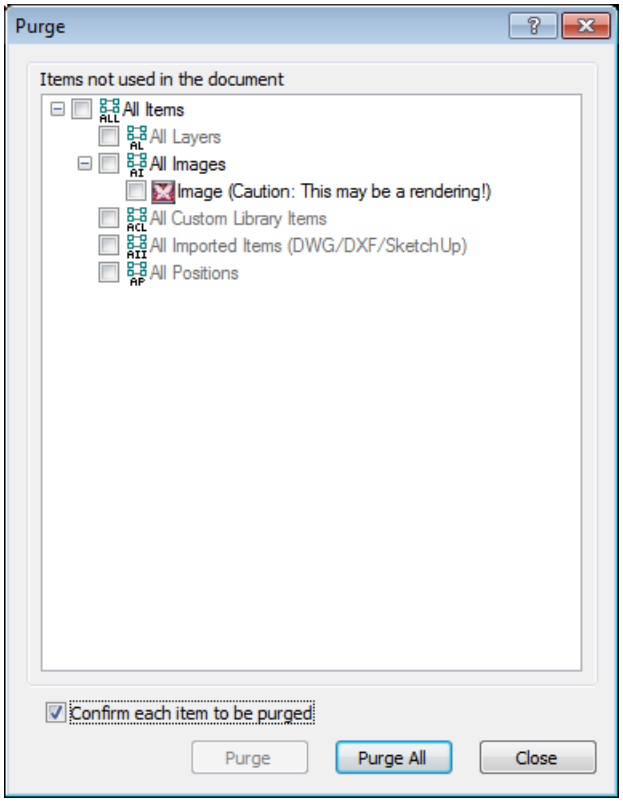
To use the Purge command

1 Click **File** > **Purge**.

2 In the warning box that appears, click Yes to proceed with the purge.

Note: Once you purge objects from your file, the action cannot be undone.

Result: The Purge window appears.



3 Click to select the items that you want to purge from your file, and then click *Purge*.

Merge command

The **Merge** command is used to combine multiple show documents into one. All file types that can be opened directly by WYSIWYG can also be merged. Therefore, an existing WYSIWYG document can be merged with

- other WYSIWYG documents
- DWG/DXF files
- Lightwright2 files
- SketchUp files

To merge WYSIWYG documents

1 Open or create the destination document.

2 From the **File** menu, choose **Merge**.

3 Click to select the file, and then click *Open*.

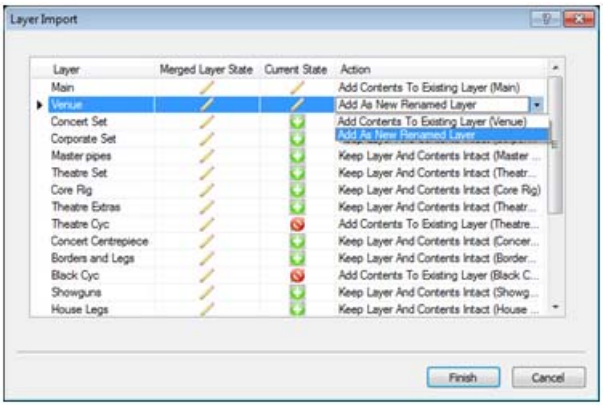
4 In the dialog box that opens, choose whether you want to Ignore Shortcuts in the file that you are merging by selecting or deselecting the checkbox.

5 Click *OK*.

Result: The *Merge > Basepoint* window appears.

6 Click to select how you want to merge the file, and then click *OK*.

Result: The Layer Import window appears. This window enables you to rename layers instead of combining them and will display the state of the layer in both files.



Tip: It is typically sufficient to accept these settings and click *Finish*. This dialog shows you which layers don't already exist and will be added, and which already exist and will have objects merged into them. By using the drop-downs in the Action column, you can rename layers instead of merging objects to your existing layers, which can help keep your file clean.

7 Click *Finish*.

8 If you chose to select the basepoint using your mouse in step 5, you must click in the document or enter the coordinates to set the basepoint.

Result: The file you selected is merged with the current file.

Note: For information on merging DWG/DXF or SketchUp documents, see “Merging a DWG/DXF document” on page 121 and “To merge SketchUp files” on page 112.

Close command

The **Close** command closes the current show document and returns you to the *Welcome* window.

To close a show document

1 From the **File** menu, choose **Close**.

2 When prompted to save changes to the current show document, click *Yes* to save changes and close the document.

Note: Click *No* to skip saving and close the document. Click *Cancel* to skip saving and keep the document open.

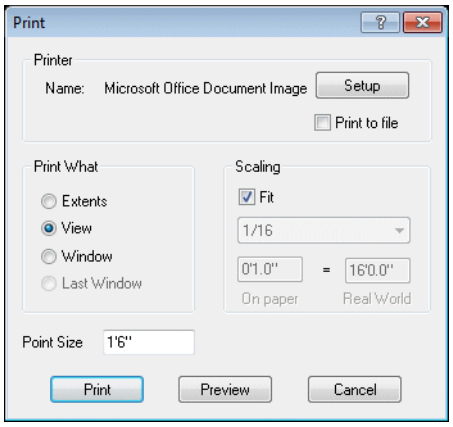
Print command

The **Print** command varies from mode to mode. In general, this command prints the active view to the selected printer.

To print a document

From the **File** menu, choose **Print**.

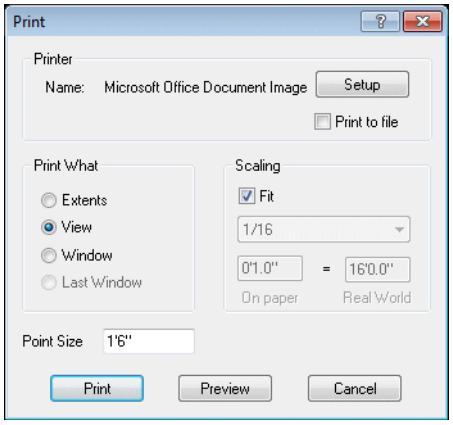
Result: Based on the mode you are in, the program may print the document directly or it may open the following dialog box so that you can set printing options:



Considerations when printing from CAD mode

Note: While you can print from CAD mode, it is strongly recommended that do not do so. Instead, please create a Layout and print it from PRES mode. For details, see “Creating a new layout” on page 436 and “To print a layout” on page 462.

The **Print** command opens the *Print* dialog box.



This dialog box displays the printer that is used to print your WYSIWYG document. The printer shown is the default printer that you have set up in Windows. Click *Setup* to change the printer or printer settings.

Select the *Print to File* check box to print your WYSIWYG document to a file instead of to a printer. When you print, WYSIWYG prompts you to enter a file name and path.

The **Print** command prints the active window using the options you specify.

Print What options

- *Extents* prints the entire drawing.
- *View* prints the currently displayed view in the active window.
- *Window* allows you to draw a window around the area you wish to print.
- *Last Window* prints the portion contained in the last window you drew for a Window print. If you have not used or previewed a Window print yet, this option is unavailable.

Scaling options

- *Fit* will print the drawing to the scale required to fill the available print area based on the selection in the *Print What* group box. When you select this check box, the drawing will print to fit.

- If the *Fit* check box is deselected, you must choose the scale from the drop-down list. Standard scales are displayed in this menu. If the scale you want to use is not displayed, choose *Custom*.
- When *Custom* is selected, you must enter the ratio. Type the ratio in the *On Paper* and *Real World* boxes. The value in the *On Paper* box is the scale ratio that you want to use on a paper copy of the document. The value in the *Real World* box is the "real" measurement for the scale ratio that you specify in the *On Paper* box. For example, you may want to use a scaling of one inch in a paper copy of your document. This one inch measurement translates to a "realistic" measurement (for example, 3 feet).

The value in the *Point Size* box is the size at which points are printed.

While it is possible to print your drawing directly from the CAD mode, it is preferable to print plots from the Presentation mode. Presentation mode includes the ability to lay out your plot with multiple views, reports, borders, and title blocks. Refer to "Printing layouts" on page 462 for more information about printing plots.

Print Preview command

Print Preview displays the document as it will look when printed. This allows you to double-check that the printed output is exactly what you want.

Print Setup command

The *Print Setup* dialog box allows you to choose the print destination, the page orientation and size, and paper tray (if applicable). The print destination may be a physical printer, or it may be a file type or fax program, for example. Paper size and source options will correspond to the printer type selected.

Recent file shortcuts

The list of recent files at the bottom of the **File** menu gives you quick access to the last four files opened. This list will continually update as show documents are opened. If the file you are looking for is not on this list, use the **Open** command.

To open a recent file

1. From the **File** menu, choose the file you want to open.
2. If another show document is currently open, you are prompted to save changes to that document before another show document is opened. Only one show document may be open at a time.

Exit command

Use the **Exit** command to exit WYSIWYG. If you have made changes to your document but have not saved yet, you will be prompted to save your changes before exiting.

To exit WYSIWYG

1. From the **File** menu, choose **Exit**.
2. When prompted to save changes to the current show document, click *Yes* to save changes and exit WYSIWYG.

   Note: Click *No* to skip saving and exit WYSIWYG. Click *Cancel* to skip saving and not exit WYSIWYG.

Options menu

Introduction

WYSIWYG offers you many options to customize the functionality of the software. You can view and set options on the **Options** menu.

The **Options** menu is the same in all the modes.

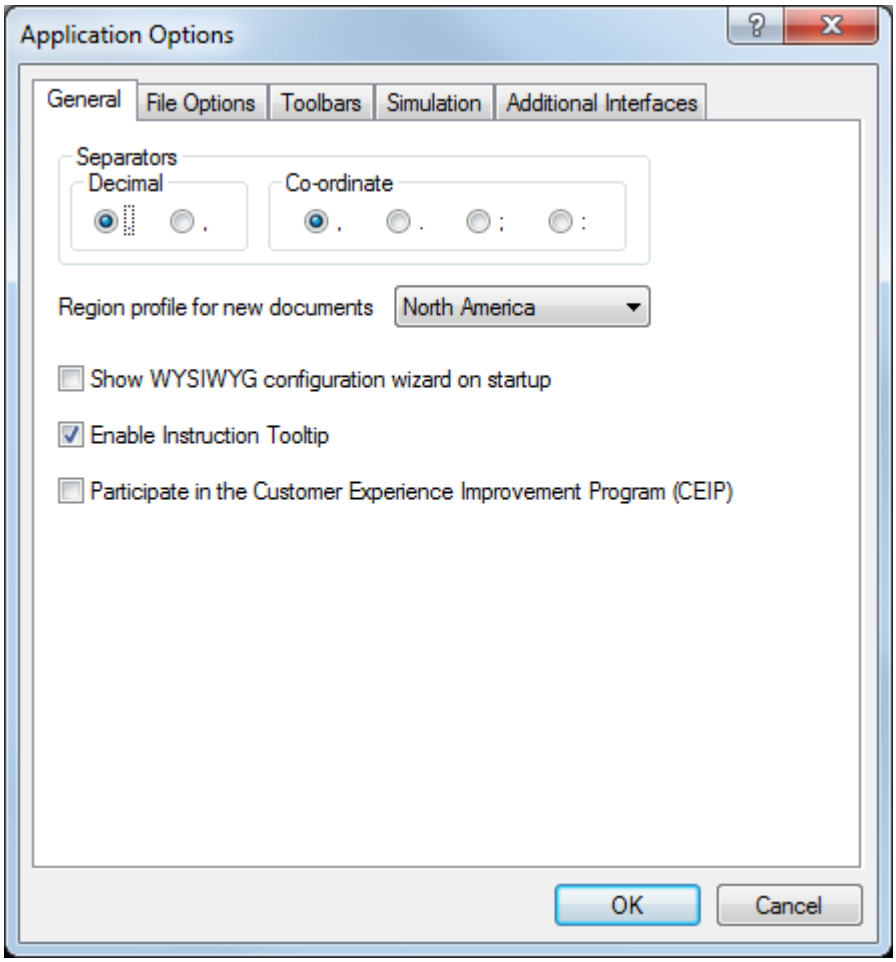
Application Options window

Application Options are general settings for the WYSIWYG software and apply to the entire application regardless of the show file that you are working on. You might want to update these settings to display specific toolbars or indicate your file-saving preferences, for example.

To modify the application options

1 From the **Options** menu, choose **Application Options**.

Result: The *Application Options* dialog box is displayed.



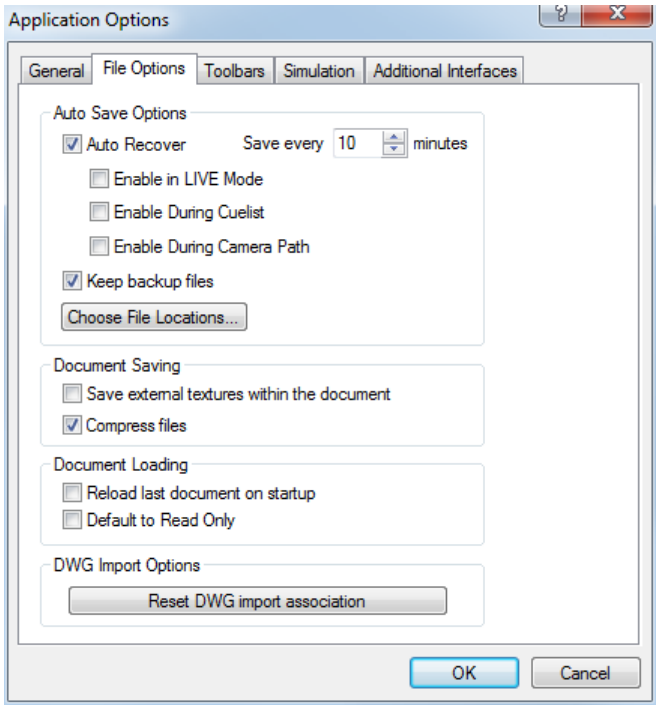
General tab

Options on the *General* tab affect the numeric separators used in WYSIWYG, and the properties for the Product Configuration Wizard.

- *Decimal*: Select the method for separating whole numbers from decimals. Available options are comma and period. Note that the decimal separator cannot be the same as the co-ordinate separator.
- *Co-ordinate*: Select the method you want to use to separate coordinates in WYSIWYG. Available options are comma, period, semicolon, colon, or slash. Note that the co-ordinate separator cannot be the same as the decimal separator.
- *Region profile for new documents*: The profile to be used for such items as voltage and which default lamp should be used in a fixture.
- *Show WYSIWYG configuration wizard on startup*: Click this check box if you want to see the product configuration wizard each time you launch WYSIWYG. The wizard enables you to choose the level of WYSIWYG that you want to run (for example, it gives you the option of launching a standard version of WYSIWYG, such as Report or Design, or a network version of WYSIWYG). If you leave this check box deselected, the same level of WYSIWYG that you chose when running the program for the first time appears by default.
- *Enable Instruction Tooltip*: Have this box checked to view helpful instruction tooltips whenever a function is performed.
- *Enable Customer Experience Improvement Program*: Have this box checked to participate in the Customer Experience Improvement Program (CEIP). Participation will enable you to contribute to the betterment of WYSIWYG. See “Customer Experience Improvement Program (CEIP)” on page 30 for more details.

File Options tab

Settings on the *File Options* tab affect your file saving settings. It is important to save your work often during your work session. WYSIWYG has an Auto Save function that will help to keep your data safe; however, it is always a good idea to get in the habit of saving your show document to an external disk periodically during your work session.



- *Auto Recover*: Select this check box to save your document at the intervals specified in the *Auto Recover save interval* box. If, for some reason, WYSIWYG closes

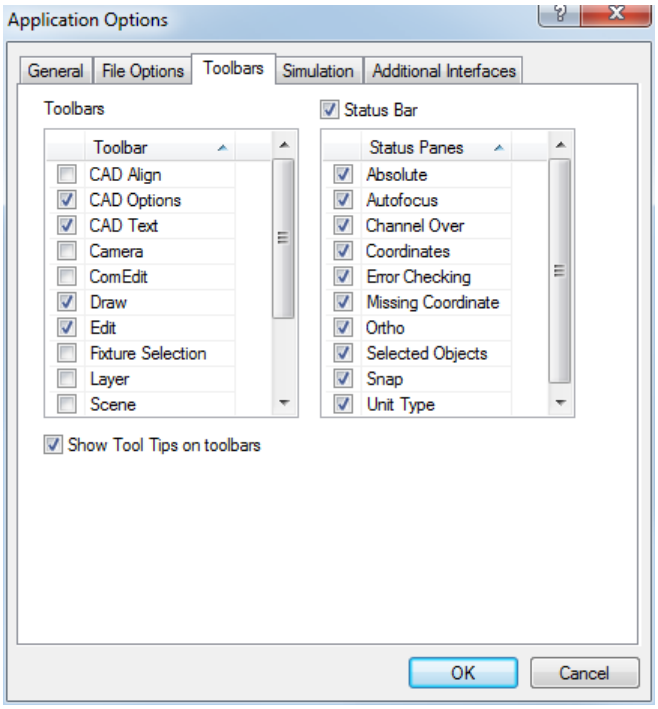
unexpectedly, such as in the case of a power loss, your document can be recovered. This recovery will only be up-to-date as of the last Auto Save, so it is recommended that you do not set this interval to be too long.

- *Enable Auto Recover in LIVE Mode*: Select this check box to enable Auto Recover while working in LIVE mode. It is recommended that you keep this option disabled since it can negatively affect performance during live simulation.
- *Keep backup files*: select this check box to save a copy of your document with the ".bak" extension every time you save the file. The backup copy is one save behind the saved copy and is saved in the same folder as the original document. Each new backup copy replaces the previous backup copy. If the main show document becomes corrupted, you can open the backup file to recover your data.

  The default settings for these options are *Auto Recover* enabled with an interval of 10 minutes and *Keep backup files* enabled.
- *Save external textures within the document*: Select this check box if you want the jpeg and/or bitmap used as textures to be saved within your WYSIWYG document. The textures will "travel" with the document and will be visible regardless of the computer on which the document is opened.
- *Compress files*: Select this check box to compress your file up to ten times its original size without affecting the contents of the file. When you select this option, files will take up less space in your computer's memory. You will also be able to send these compressed files as email attachments to your partners and clients with greater ease.
- *Reload last document on startup:* Select this check box to ensure that the last document that you have been working on in WYSIWYG is loaded automatically when you launch WYSIWYG.
- *Default to Read Only*: Select this check box to load documents in read-only format. This format is helpful if you are loading someone else's document and you do not want to accidentally make changes to it.
- *Reset DWG import association*: Click this button to reset the association of specific blocks in DWG files with certain pre-selected fixtures. When you import DWG files, one of the options is to associate a block that is found in a DWG file with a specific fixture. If you decide to remember this association for future imports, all blocks with that name will be imported as the chosen fixture.

Toolbars tab

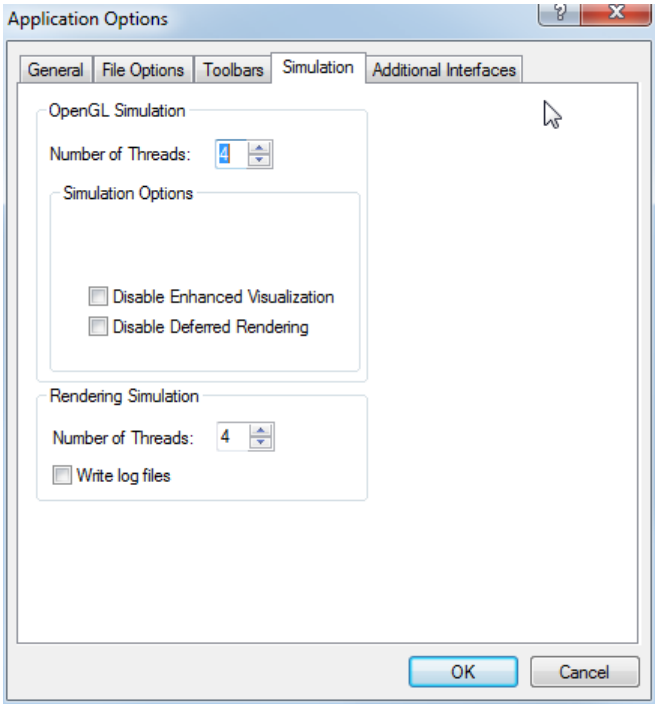
Options on the *Toolbars* tab affect which toolbars and status bar panes you want to display in WYSIWYG, and whether you want to display tooltips.



You can also hide the tooltips that display information about a specific fixture, hang structure, or object when you hover over the fixture. Tooltips are displayed by default. For more information on toolbars, refer to “Toolbars” on page 59.

Simulation tab

Options on the *Simulation* tab affect the type of simulation that you want to use, the processing power for beam simulation, and the memory and processing power available for rendering.

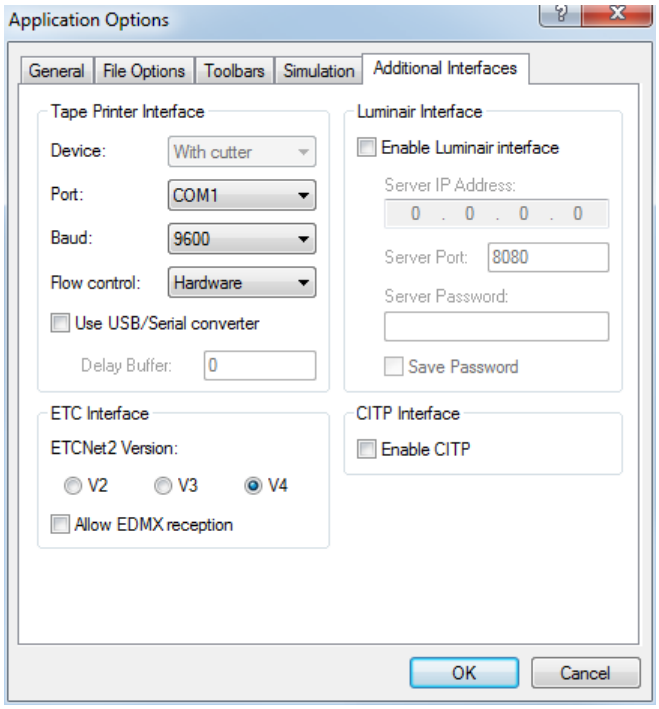


- *OpenGL Simulation - Number of Threads*: Select the number of processors that you have available on your computer for beam simulation. Modify this number only if you upgrade the number of processors.

- *Simulation Options - Disable Enhanced Visualization*: Select this option if your video card does not support enhanced features for beam simulation. This option disables enhanced visualization features. Once you select this value, some advanced visualization features will turn off in the visualization and revert to a simplified Enhanced Beam visualization. For more information see see “Visualization Requirements” on page 21.
  Note: Switching to another non-Shaded view, then returning back may be required to allow this change to take effect.
- *Simulation Options - Disable Deferred Rendering*: Deferred Rendering provides a performance boost to WYSIWYG's Shaded Views. Check to disable Deferred Rendering for all Shaded Views whenever WYSIWYG is run. On some systems, deferred rendering is not supported and will be automatically disabled. For more information see see “Visualization Requirements” on page 21.
- *Rendering Simulation - Number of Threads*: This box is auto-filled with the number of processors and/or processor cores available in your computer. It is used to specify the number of processors available for rendering. Modify this number only if you want WYSIWYG's renderer to use less than the total number of processors/ processor cores available in your computer.
- *Write log files*: Select this check box to generate a log file during rendering. Log files are stored in the "Render" folder, which is located inside the "Bin" folder within the WYSIWYG installation folder. The typical path to this folder on some operating systems (Windows Vista, 7, 8) is C:\ProgramData\CAST Software\WYSIWYG\<Release folder>. Or, on a Windows XP system it is C:\AppData\CAST Software\WYSIWYG\<Release folder>.

Additional Interfaces tab

Options on the *Additional Interfaces* tab affect the options that will allow you to monitor EDMX data directly, the pipe tape printer, as well as the Luminair and CITP interfaces. Direct EDMX reception allows WYSIWYG Perform users to monitor EDMX data without the need to connect to a console with the Device Manager.



Tape Printer Interface

- *Device*: Specify whether or not the pipe tape printer has a tape cutter.
- *Port*: Specify which port on your computer the pipe tape printer is connected to.
- *Baud*: Specify the baud rate at which the pipe tape printer is running. The baud rate is the rate at which your modem can transmit and receive data.
- *Flow Control*: Select the method of data control between the pipe tape printer and the machine. Available options are Hardware, Software, and None. The default is Hardware.
- *Use USB/Serial converter*: Check this option if your pipe tape printer is connected through a USB/Serial converter. (This is required if a Serial port is not available on your computer).
- *Delay buffer*: Set the delay buffer (in milliseconds) for the USB/Serial converter here. Check the documentation that came with your converter for details. The default is 0, which should be acceptable in most cases.

Note: Be sure to check in the Device Manager that these parameters match those of the COM port to which the Pipe Tape Printer is connected.

Luminair Interface

Notes:

- For information on how to set up and use the Luminair connectivity, see “Connecting to Luminair” on page 581.
- This option is only available with WYSIWYG Design and WYSIWYG Perform

- *Enable Luminair interface:* Select this option if you want to transfer Fixture Point of View (PoV) images to the Luminair app on the iPhone or iPod Touch.
- *Server IP Address*: Enter the IP address of the iPhone/iPod Touch to which you will be transferring images.
- *Server Port*: The port used by the Luminair file server is 8080, a value that cannot be changed.
- *Server Password*: Type the password of the Luminair file server.
- *Save Password*: Click this check box to encrypt the Luminair file server password and save it locally so that you do not need to re-enter it when you restart WYSIWYG.

ETC Interface

Note: This option is available only with WYSIWYG Perform.

- *ETCNet2Version*: Select which ETCNet2 network version you want to monitor. You can monitor only one version at a time.
- *Allow EDMX reception*: Select this check box to enable direct EDMX reception in WYSIWYG.

CITP Interface

Note: This option is available only with WYSIWYG Perform.

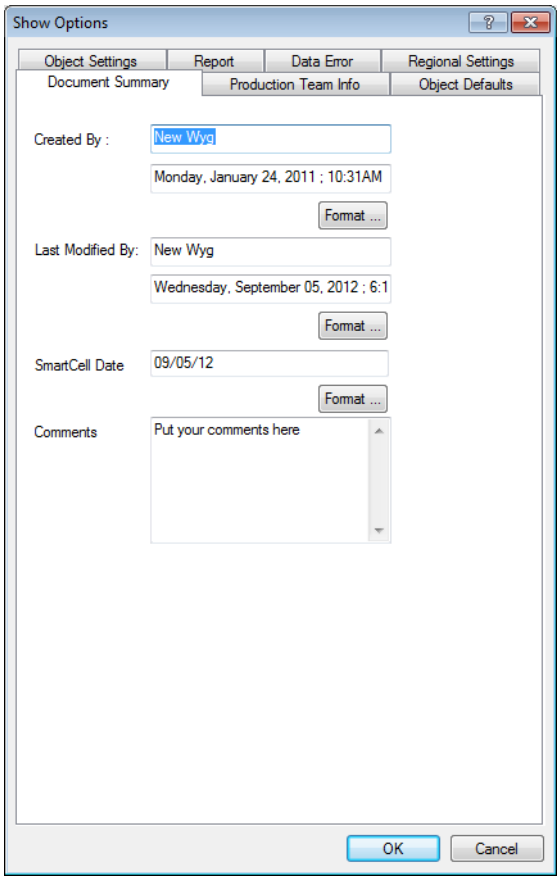
- *Enable CITP*: Check this box to enable CITP data reception, and then restart WYSIWYG. For more information about working with CITP, see “To configure a new video source for streaming video” on page 341.

Show Options window

Show settings that are specific to the current show file, such as show name, designer, and venue, are set in the Show Options dialog box. You might want to modify these settings when you start a new show file. Error conditions and the appearance of certain objects in your show file are also specified in Show Options. To open the Show Options dialog box, from the **Options** menu, click **Show Options**.

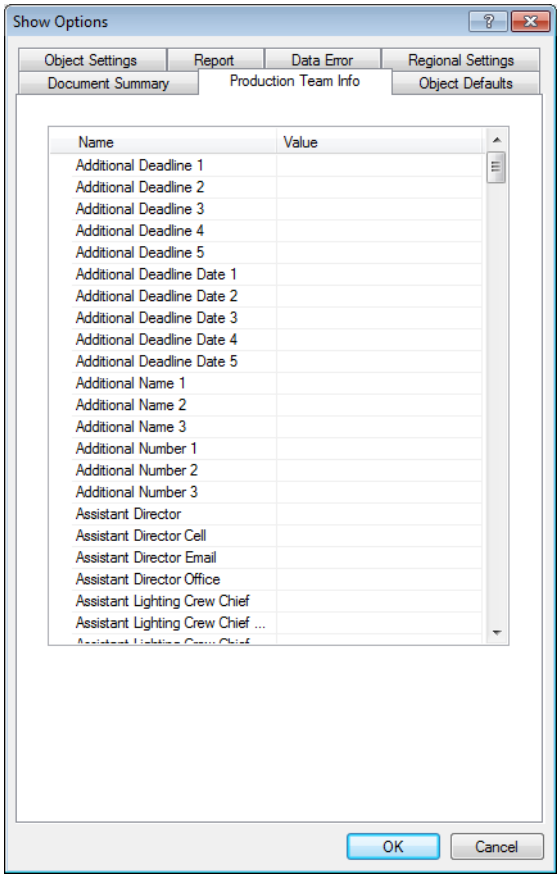
Document Summary tab

Options on the *Document Summary* tab affect the document information (such as name of the person who created the document and the name of the person who last modified the document). The date stamps are not editable, but will track the creation and modification dates of the file.



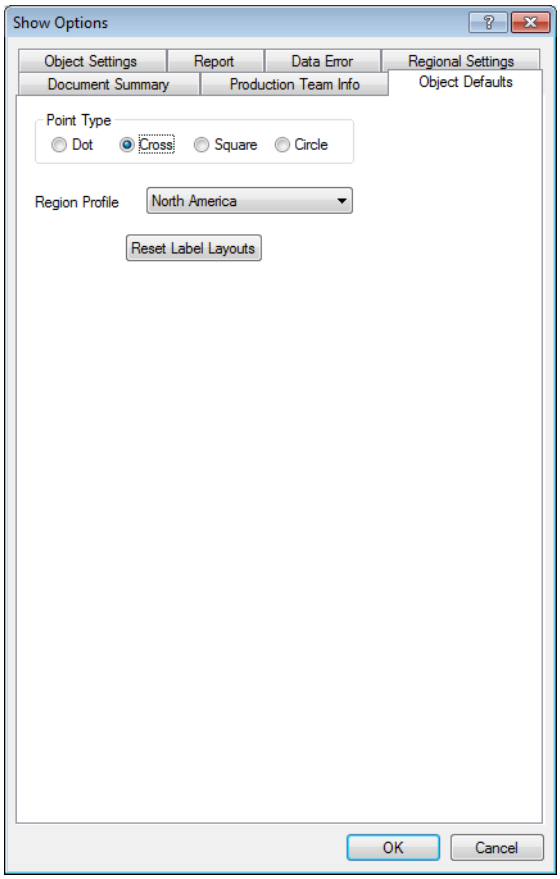
Production Team Info tab

This tab shows all of your production team's information in one easy to access place. The values in this tab reflect those that you enter in the Production Team Info Wizard. For more information on this Wizard, see "Production Team Info Wizard" on page 469.



Object Defaults tab

Options on the *Object Defaults* tab affect the default settings for the regional profile. The profile that you choose determines items such as the type of bulb to use in a particular fixture. This also sets how points will be displayed (as crosses, dots, squares, or circles).

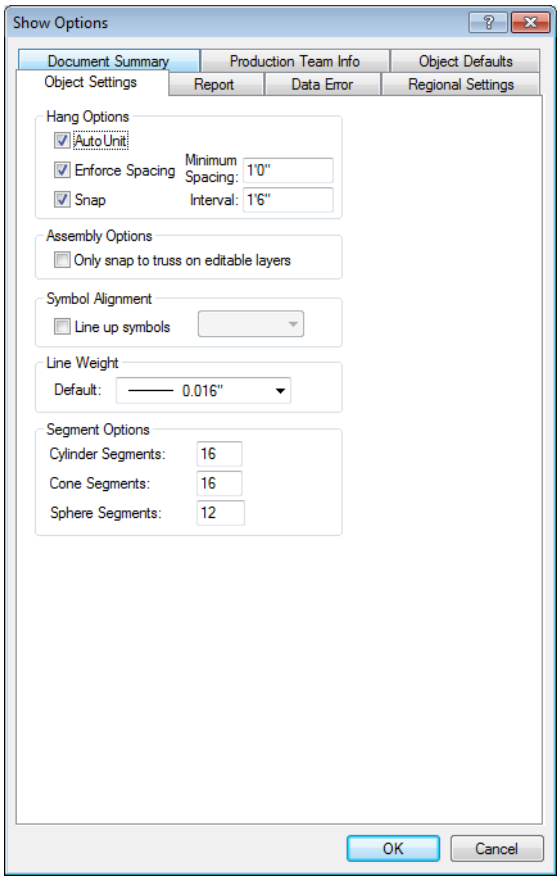


Point type: Indicates the default point type in your show document. You can choose to display points as dots, crosses, squares, or circles.

- *Region profile*: The geographical region where the show will be occurring. This setting establishes the profile for voltage and consequently, what will be used for such items as light bulbs in a fixture.
- *Reset Label Layouts:* Click to reset all fixture labelling to the default settings.
- *Set Default Font for Presentation Views*: Click to select a default font for presentation views.

Object Settings tab

Options on the *Object Settings* tab affect how various objects are placed or drawn in CAD mode.

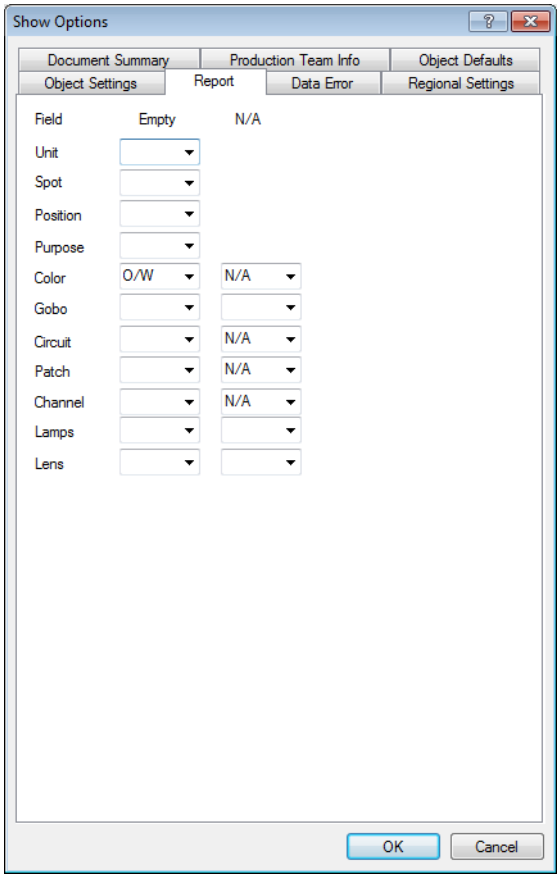


- *AutoUnit:* Select this check box to automatically assign a unit number to each fixture that is hung. This option is available for pipes only. It does not apply to truss.
- *Enforce Spacing*: Select this check box to indicate that the spacing you specify in the *Minimum Spacing* box is enforced.
- *Snap*: Select this check box to enable pipe snap. Pipe snap places fixtures on a pipe at a specific interval as specified in the *Interval* box. For information on snaps, refer to “Pipe snap” on page 279.
- *Minimum Spacing:* Type the minimum spacing between fixtures. This option prevents you from hanging the fixtures too close to one another. The minimum spacing cannot be greater than the value you specify in the *Interval* box.
- *Interval*: Type the distance between fixtures. This option is used for pipe snaps.
- *Only snap to truss on editable layers*: When selected, CAD snaps will only snap to truss on editable layers; all truss objects on un-editable layers will be ignored by CAD snaps.
- *Line up symbols:* Select this check box to line up fixture symbols at specific angles, and then select an increment from the drop-down list. Symbols will shift from their focused position to the nearest increment of the chosen angle. For example, symbols will be drawn in one of four directions when the increment is set to 90 degrees. This setting does not affect the focus of the beam; rather it is intended for the “cleanliness” of the printed plot.
- *Line Weight - Default*: Type the default weight of a line when it is printed or viewed using print preview.
- *Cylinder Segments*: Type the number of segments into which you want to break a cylinder.
- *Cone Segments*: Type the number of segments into which you want to break a cone.

- *Sphere Segments*: Type the number of segments into which you want to break a sphere.

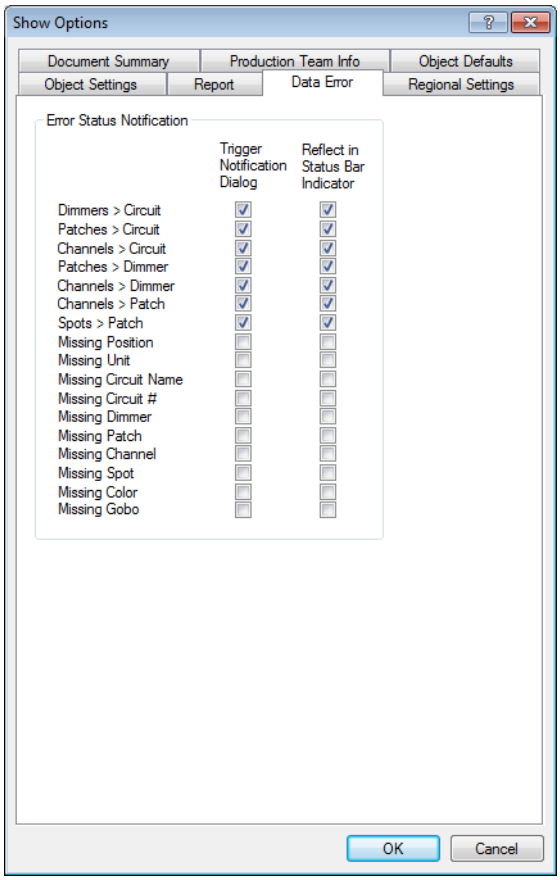
Report tab

Options on the *Report* tab affect the default values for empty or non-applicable cells in spreadsheets.



Data Error tab

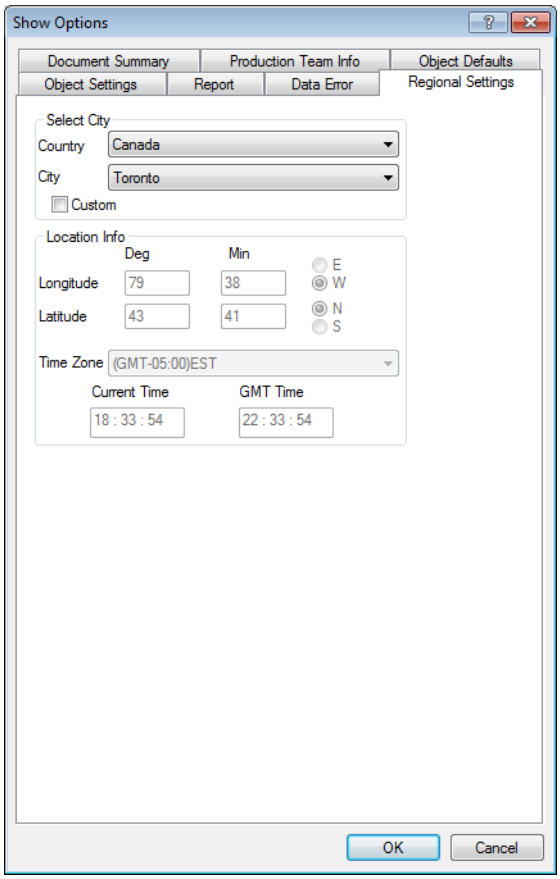
Options on the *Data Error* tab control what types of data errors trigger notification boxes and status bar notifications.



For more information on data errors, refer to “Errors” on page 376.

Regional settings tab

Options on the *Regional settings* tab set the geographic location of your show. You can either select the country and city or enter the latitude and longitude of the location. These settings are used in Shaded views and renderings that use environmental options.



- *Custom*: Select this check box to enter the latitude and longitude values in the *Latitude* and *Longitude* boxes, respectively. Click to clear this check box to select a country and city from the *Country* and *City* drop-down lists, respectively.
- *Time Zone*: After selecting *Custom*, select the time zone for your show. When you select a time zone, the current time as set on your computer is displayed in the *Current Time* box. The GMT value is also shown.

Notes:

- To ensure accuracy in Time of Day simulations, you must define the North direction in your document. For more information on defining the North direction, see “Drawing a compass” on page 170.
- The values that you enter on this tab are reflected in Step 6 of the Render Wizard. For more information on the Render Wizard, refer to “Rendering” on page 547.
- The values that you enter on this tab are also reflected in the Shaded view options' Time of Day tab. For more information on the Shaded view options, refer to “Modifying shaded views” on page 81.

View Options dialog box

View options set how objects are drawn and displayed in Wireframe and shaded views.

For more information on View options for Wireframe views, refer to “Modifying wireframe views” on page 73.

For more information on View options for shaded views, refer to “Modifying shaded views” on page 81.

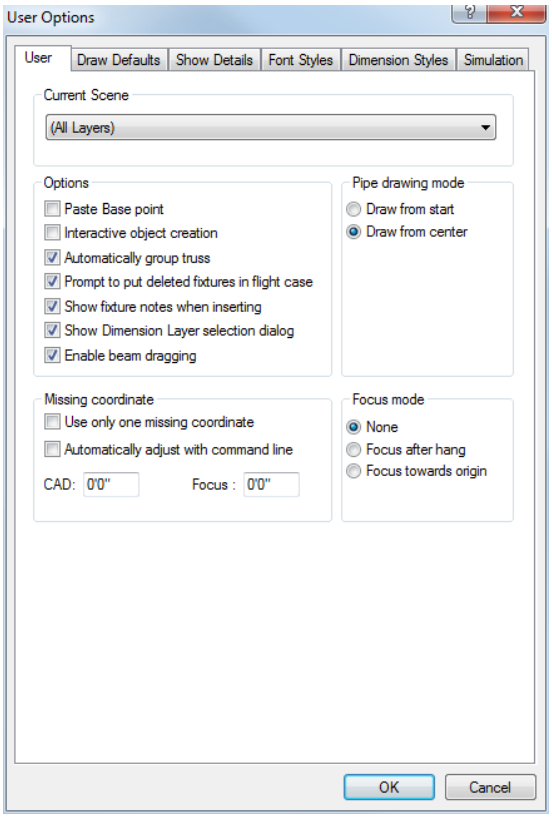
User Options dialog box

The User Options dialog box enables you to set up your working preferences. You might, for example, want to specify how the snap operation will work or how much information will be displayed on your plot. The settings that you choose are in effect for the entire time that you are working in WYSIWYG regardless of the show file that you are working on.

To open the User Options dialog box, click **Options** > **User Options**.

User tab

Options on the *User* tab affect the current scene selection, options, pipe drawing mode, missing co-ordinate, and focus mode settings.

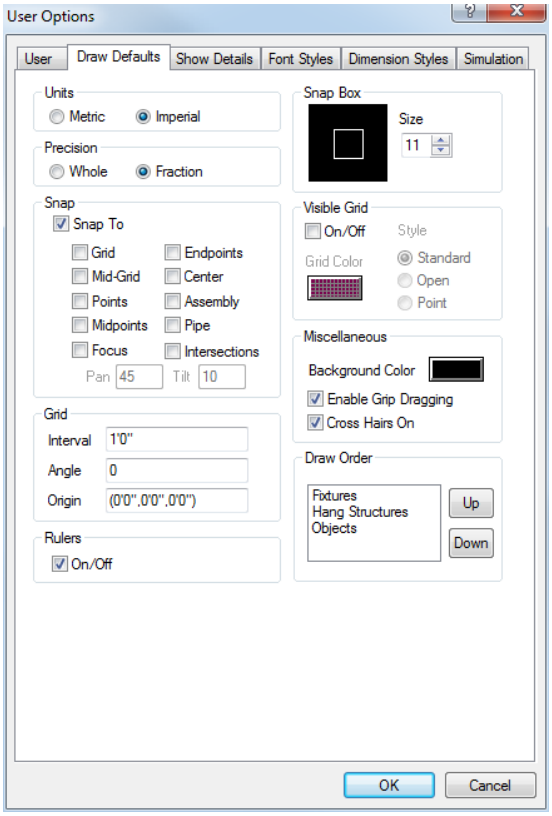


- *Current Scene:* The current layer shown on the plot. The current scene can also be changed on the Scene toolbar.
- *Paste Base point*: Select this check box to enter a base point before an object is copied or cut. This will be the object’s reference point for paste commands. Click to clear this check box to allow the base point to be the insertion point of the object.
- *Interactive object creation:* Select this check box to allow yourself to draw objects using the mouse instead of entering values in dialog boxes.

- *Automatically group truss:* Select this check box to group truss objects together in a similar manner to choosing **Group** from the **Edit** menu. This option is helpful in that it treats the truss as a single unit and allows you to perform actions more effectively on multiple objects. If you want to edit one truss object, you must first use the Ungroup command on the **Edit** menu to remove the grouping from the truss objects.
- *Prompt to put deleted fixtures in flight case*. Select this check box to be prompted when you delete a fixture. You will be prompted to delete it completely or send it to the Flight Case. If you choose to send it to the Flight Case, the fixture is reserved for you in case you want to reinsert it.
- *Show fixture notes when inserting*. Select this check box to display the fixture notes when you are hanging fixtures, if applicable to the selected fixture.
- *Enable beam dragging*: Select to enable click and drag positioning of beams in CAD, Design, and Live modes. This option is also available on the CAD Options toolbar in CAD, Design, and Live modes.
- *Missing coordinate*: Sets missing coordinate options. For more information on missing coordinates, refer to “The missing coordinate” on page 102.
- *Pipe drawing mode:* Sets the preference for drawing pipes (either from the start or from the center). For more information on drawing pipes, refer to “Drawing pipes” on page 162.
- *Focus mode:* Allows you to choose to focus a fixture as you hang it in your drawing. Specify one of the following options to set how a fixture will react after you hang it in a plot:
  - *None.* This setting indicates that you can continue to hang fixtures uninterrupted.
  - *Focus after hang*. If enabled, you will be prompted to focus each fixture as you hang it. Once the fixture is focused, you can continue hanging other fixtures.
  - *Focus towards origin*. If enabled, the fixtures that you hang are automatically focused towards the user origin as you hang them. You can continue to hang other fixtures uninterrupted.

Draw Defaults tab

Options on the *Draw Defaults* tab affect general preferences for snap, grid, and units settings.



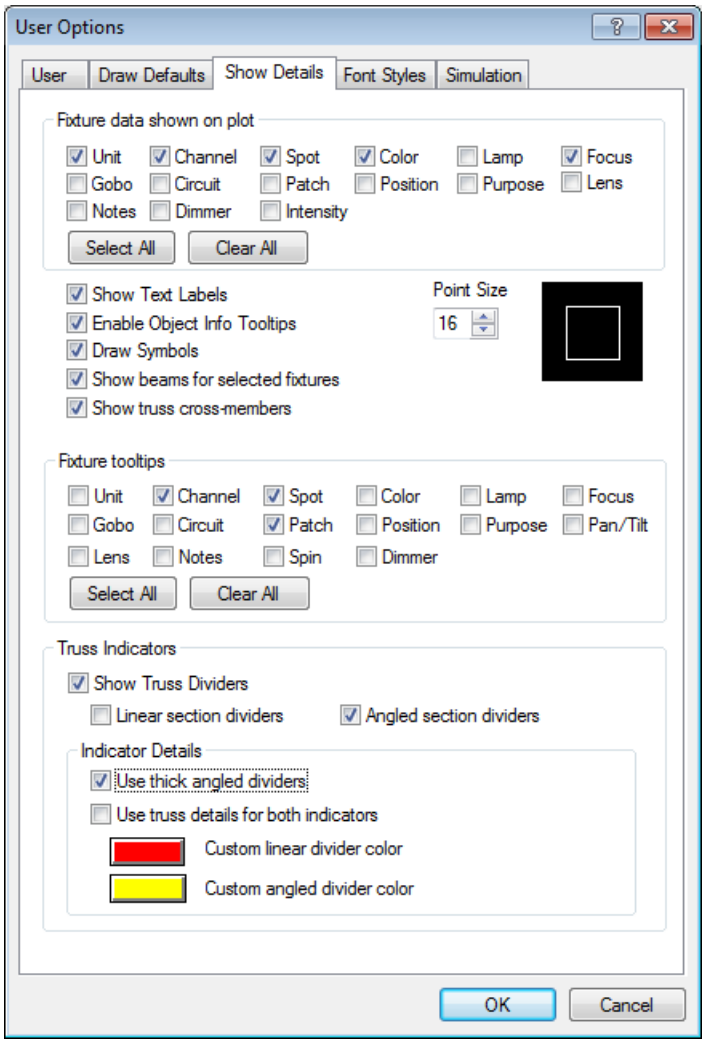
- *Units of measure*: For more information on these units, refer to “Metric vs. Imperial” on page 99. The units can be toggled from metric to imperial at any time by double-clicking on the units display on the status bar.
- *Precision*: affects the units of drawing precision. For more information on these units, refer to “Coordinate notation” on page 98. Based on your choice for measurement units, you can choose to display the drawing precision in centimeters, millimeters, whole numbers, or fractions.
- *Snap To*: Enables or disables snap. For more information, refer to “Snaps” on page 105. Snap tools can be toggled at any time using the tools on the CAD Options toolbar.
- *Interval*: Select the spacing between points on the drawing grid. Although the WYSIWYG drawing grid might not be visible, it exists in the background. The settings in this group box are important when using the snap to grid tool. For more information on this tool, refer to “Snaps” on page 105.
- *Angle*: Select the angle of rotation for the drawing grid. Although the WYSIWYG drawing grid might not be visible, it exists in the background. The settings in this group box are important when using the snap to grid tool. For more information on this tool, refer to “Snaps” on page 105.
- *Origin*: The point of origin. For more information on the origin, refer to “Coordinate system and origin” on page 98.
- *Rulers*: Click this check box to show or hide the rulers in drawing modes.
- *SnapBox Size:* Sets the size of the snap box that appears around the cursor when in snap mode. Consequently, this affects how close the cursor must be to the objects before the snap is applied.
- *Visible Grid*: Select the *On/Off* check box if you want to add a grid to your wireframe view (in every view except isometric). Then click the *Grid Color* box to choose the

color for the grid lines. You can also choose the style of the grid by selecting the appropriate option button. *Standard* creates a grid with evenly spaced, closed squares. *Open* creates a grid with open squares. *Point* creates a grid of dots. The lines in the resulting grid are spaced at the interval specified in the *Interval* box. If the interval is short, you may have to zoom in on your plot to see the gridlines.

- *Background Color:* The color that will be used in all wireframe views. Click the box to choose a different color.
- *Enable Grip Dragging:* Select this check box if you want to be able to resize objects by clicking and dragging on their markers.
- *Cross Hairs On:* Select this check box to set the cursor display to use cross hairs in plot views. Cross hairs are useful for lining up objects.
- *Draw Order*: Arrange the order in which fixtures, hang structures, and objects will be drawn in your model by selecting the items and moving them up or down with the arrow buttons. Whichever item appears at the top of this box will always appear on top of the other items in your model. For example, if you leave the default order of Fixtures > Hang Structures > Objects, then fixtures will always appear on top of truss and truss will always appear on top of all other objects, regardless of how you arrange them via the Send to Back and/or Bring to Front commands. For more information on these commands, see “Sending to Back/ Bringing to Front” on page 193.

Show Details tab

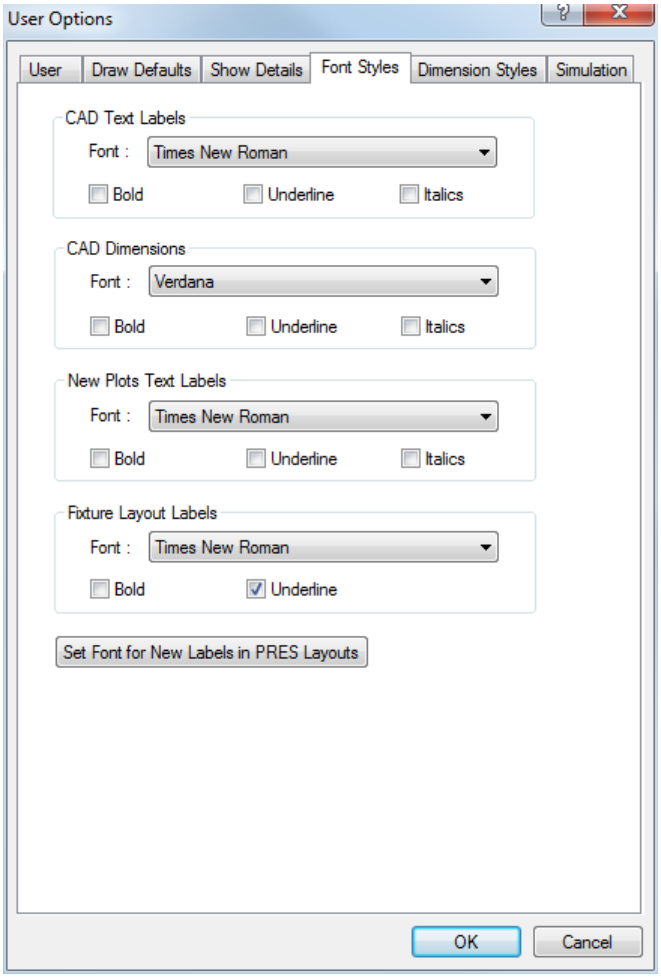
Options on the *Show Details* tab affect the amount of information that is shown on your drawing. This includes how much data is shown on the plot.



- *Fixture data shown on plot:* Indicates the fixture data that you want to display on your drawing.
- *Show Text Labels*: Select this check box to display text labels.
- *Enable Tooltips:* Displays tooltips on your drawing. Tooltips are the pop-up messages that display specific information about a fixture when you hover over it. Choose the information that you want to display in a tooltip by selecting the appropriate check boxes in the *Fixture tooltips* group box. Tooltips are often helpful when trying to locate a specific fixture in a plot.
- *Draw Symbols:* Select this check box to draw fixtures as symbols rather than as 3D objects. For more information on symbols, refer to “Symbols” on page 308.
- *Show beams for selected fixtures:* Select this check box to automatically turn on beams when selecting a fixture on your drawing. If you do not select this box, the beam is not visible when you select a fixture.
- *Show truss cross-members*: Click to clear this check box to hide truss cross members on your drawing to reduce the clutter or to increase the speed of the display. Select this check box to see the cross members.
- *Point Size:* Select the default size for all points. You can view a sample of the chosen size in the box to the right.
- *Fixture Tooltips*: Select which fixture properties are displayed in the tooltips. This setting can be useful when looking for a specific fixture in a plot.
- *Truss Indicators* > *Show Truss Dividers*: Truss Indicators show where the breaks in your truss sections are for quick assembly on site and ease-of-use for rental shops and other personnel who need to analyze your truss configuration. Click *Show Truss Dividers* to enable this feature, and then select the options below. For more details on using this feature, see “Configuring truss indicators” on page 265.
- *Linear Section Dividers* – Click to show the lines that appear parallel to the ends of your truss sections. These lines are always thick and can be colored any way you choose.
- *Angled Section Dividers* – Click to show the lines that appear at the junction of truss sections on a 45 degree angle with the truss end. The Angled Section dividers can be thick or thin, and can be colored any way you choose.
- *Use thick angle dividers*: Select this option to display thicker angled section dividers.
- *Use truss details for both indicators*: Click this option if you want the linear and angled dividers to be shown using the colour and line weight settings of each truss. To give dividers a uniform and distinct look, deselect this option, and then click the colour boxes to choose the appropriate colour for each indicator.

Font Styles tab

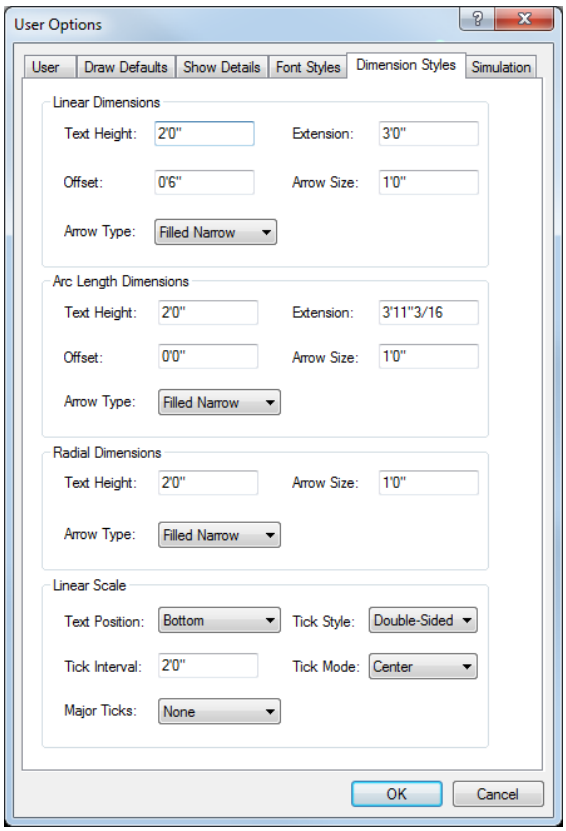
Options on the Font Styles tab enable you to define the default font and font style for CAD text labels and dimensions, New Plots text labels, and Fixture Layout labels.



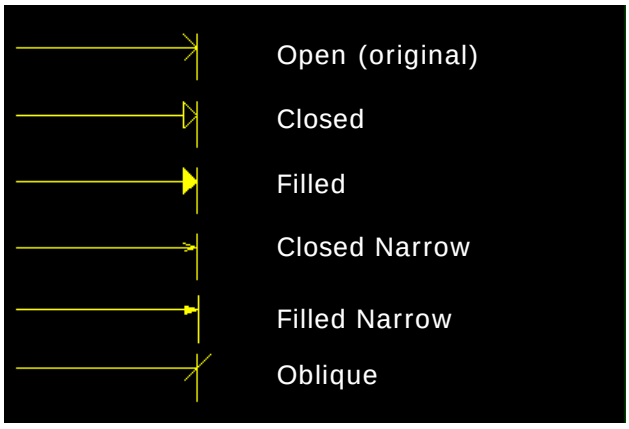
- *CAD Text Labels*: Select the default font for text labels and focus position labels in CAD mode, and then select the desired text formatting options (bold, italic, or underlined).
- *CAD Dimensions*: Select the default font for dimension objects in CAD mode, and then select the desired text formatting options (bold, italic, or underlined).
- *New Plots Text Labels*: Select the default font for text labels in New Plots, PRES mode, and then select the desired text formatting options (bold, italic, or underlined).
- *Fixture Layout Labels*: Select the default font for all fixture Layout labels, and then select the desired text formatting options (bold or underlined).
- *Set Font for New Labels in PRES Layouts*: Click to select default font options for new labels in PRES mode Layouts. You can choose options such as the font, font style, size, color, and script.

Dimension Styles tab

Options on the Dimension Styles tab enable you to customize all aspects of dimension objects. Here the properties of linear, arc length and radial dimensions can be set. Once the settings have been chosen, all new dimension objects inserted in the file will use these properties.



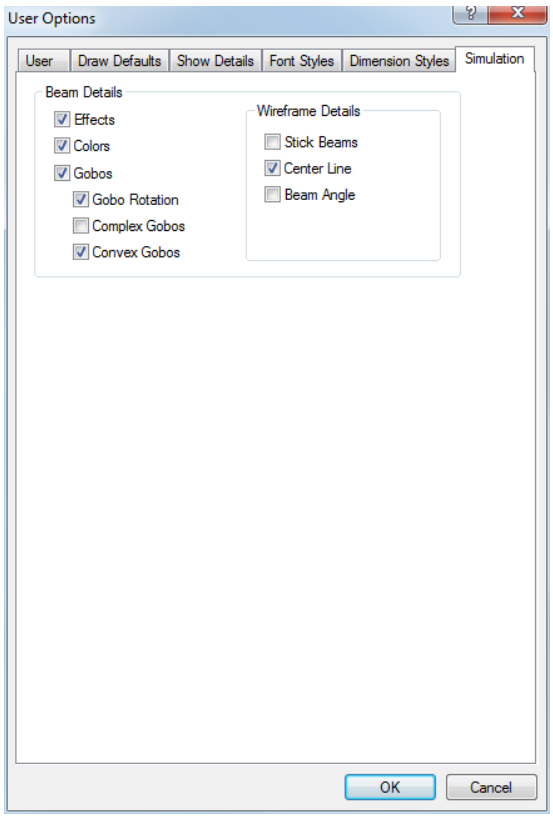
- *Text Height*: Sets the default text height for dimension objects in CAD mode.
- *Extension*: Sets the default length of the extension lines for dimension objects in CAD mode.
- *Offset*: Sets the default offset value for dimension objects in CAD mode. The offset is the distance between the extension lines and the object being measured
- *Arrow Size*: Select the default size of the arrow head for dimension objects in CAD mode.
- *Arrow Type*: Select the default arrow type for dimension objects in CAD mode. There are several styles available to choose from.



- *Text Position*: Select the default position of text that appears for linear scales.
- *Tick Style*: Select the default tick style that will appear on linear scales.
- *Tick Interval*: Select the default interval at which ticks will appear on linear scales.
- *Tick Mode*: Select the default mode of which ticks appear on linear scales.
- *Major Ticks*: Select how major ticks will appear on linear scales.

Simulation tab

Options on the *Simulation* tab are only set for WYSIWYG Design and Perform. These settings determine how detailed simulation views are, and can affect the speed of beam simulation in simulation views.



For more information on simulation options, refer to “Visualization” on page 541.



WYSIWYG Viewer

The WYSIWYG Viewer is a separate application that allows non-WYSIWYG users to view and print your WYSIWYG (.wyg) files. It is beneficial to those individuals who are interested in viewing or printing a WYSIWYG file but who do not own a licensed copy of WYSIWYG.

Any file with a .wyg file extension can be viewed and printed from the WYSIWYG Viewer.

WYSIWYG Viewer comes with a Shaded view so you can see a realistic, 3D representation of the file, as well as Presentation mode. To ensure that every aspect of your file can be seen properly, ensure that all relevant information is available in Presentation mode before opening your file in the WYSIWYG Viewer.

All functionality except navigation and printing has been disabled. Users cannot edit files, nor can they insert items into a file. For more information on preparing your file in Presentation mode, refer to Chapter 8, “Presentation mode.”

The WYSIWYG Viewer is available from the CAST Software Web site at http://www.cast-soft.com/wysiwyg/.

Chapter 8

Presentation mode



Presentation (Pres) mode contains all the necessary tools for creating professional printouts of your show document, including reports, plots, and images.

In this chapter

Layout tabs	413
Reports	414
New Plots	418
Layouts	436
Images	463
Worksheets	464
Pipe tapes	467
Production Team Info Wizard	469



Layout tabs

Introduction

Beneath the work area in each mode is a series of layout tabs. These tabs provide various configurations of the views that you are working with. To change layouts, click the tab that corresponds to the layout that you want to use. Presentation mode contains the following layout tabs:

- Reports - The work area displays reports formatted for printing.
- New Plots - The work area displays plots
- Layouts - The work area displays layouts formatted for printing.
- Images - The work area displays bitmap or jpeg images that are stored with the show file.
- Worksheet - The work area contains spreadsheets that the user can edit.

Reports

Introduction

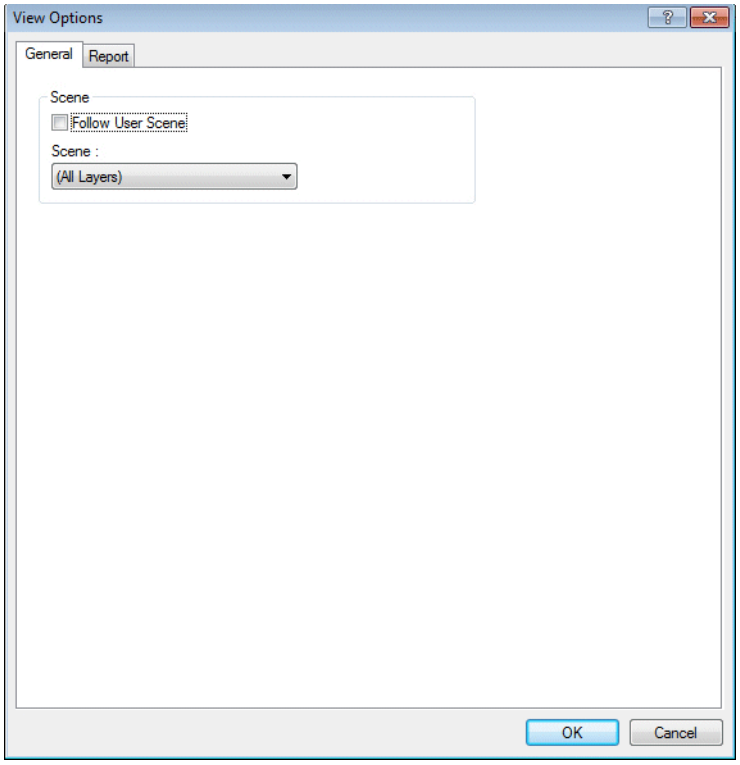
The Reports layout tab opens a series of pre-formatted reports that are generated by WYSIWYG, containing all data entered in CAD and Data modes. These reports are available for you to use as is, or to modify to suit your needs. Unlike the spreadsheets in Data mode, you cannot edit fixture data here. Reports are designed to be printed documents. You can, however, modify their setup and layout.

Working with reports

To modify a report

1 From the **Options** menu, choose **View Options**.

Result: The *View Options* window is displayed.



Tip: You can use the *View Options* tool on the *Standard* toolbar. 

2 Modify options as desired.

3 Click *OK*.

General tab

Options on the *General* tab affect the scene that is displayed in the report.

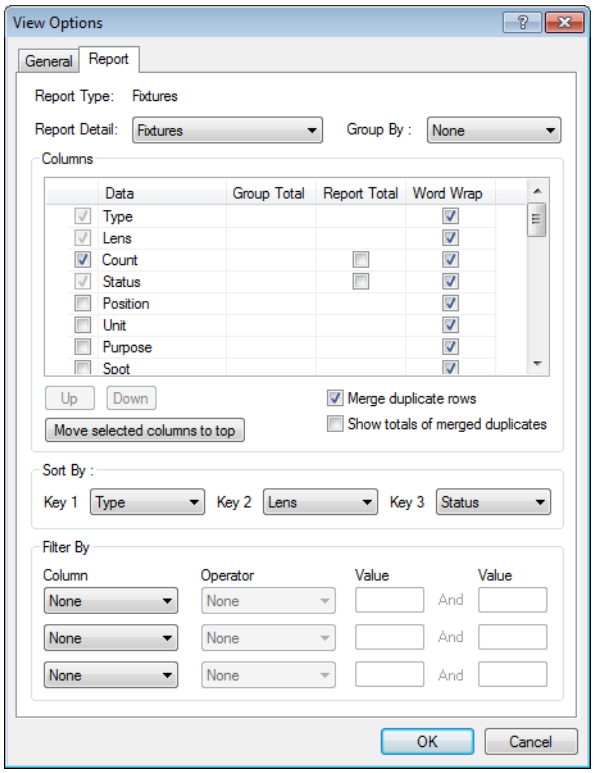
- *Follow User Scene*: Select this check box to filter the report according to the current scene. The current scene is displayed in the drop-down list on the *Scene* toolbar and in the *Scene Database* dialog box (from the **File** menu, choose **Scenes**).

To filter your show information based on the scene selected, click to clear the check box, and then select the desired scene from the *Scene* drop-down list. Only fixtures on the active layers (or layers in the scene) are displayed in the report.

Tip: You can change the current scene at any time by using either the *Scene* toolbar or the *Scene Database* dialog box. For more information on scenes, refer to “Scenes” on page 223.

Report tab

Options on the *Report* tab affect the appearance of the report.



- *Report Type*: Indicates the type of report. Can be either one of Fixtures or Truss. The Report Type determines the available options under Report Detail and the columns available for use in the report.
- *Report Detail*: Changing the detail level will change how the row information is displayed in your report.
  - **Example 1**: Given a Three Cell Cyc light, the Fixtures detail level will only give you one row of data: Three Cell Cyc. If you switch the detail level to Circuits, then you will get three rows of Three Cell Cyc, one for each circuit.
  - **Example 2**: Given a Source 4 with a Scroller attachment, a report with the Fixtures detail level will simply display “Source 4”. The same report with the Fixtures and Accessories detail level will display one row for Source 4 and one row for the scroller attachment.
- *Group By*: Select a data field for grouping data. These groups will appear as separate tabs at the bottom of a report. In the case of an Instrument Schedule, fixtures are grouped by hanging position. Each position appears on a separate page.
- *Columns*: Each row in the Columns table represents a column that can be displayed in the Report. To enable a header, enable the checkbox on the row of the relevant data column.
- *Columns – Group Total*: This checkbox appears only when Group By is not set to None. When selected, WYSIWYG generates the sum-total of the data column underneath the last cell in that column.

- *Columns – Report Total*: When Group By is set to None, Group Total will generate a sum-total of the data column underneath the last cell in that column; otherwise, a new Totals tab will be created, listing the sum-total of the data for that column across all groups.
- *Columns – Word Wrap*: When selected, the text in that column will use word wrap to try to fit all the text in the cell.
- *Up/Down*: These controls let you re-order the way columns appear in your report. First, select a row in the Columns table, and re-arrange it using Up/Down buttons.
- *Move selected columns to top* – When selected, all visible columns will be shifted to the top of the Columns table.
- *Merge duplicate rows* – When selected, all rows with duplicate Type names will be grouped into one row. For example, a report that has a Three Cell Cyc with Circuits Report Detail will show three individual rows, one for each circuit in the Three cell Cyc. With merge duplicate rows enabled, the three circuits will be combined into one row.
- *Show totals of merged duplicates* – This option is enabled only when *Merge duplicate rows* is enabled. After merging duplicate lines, this gives you the option to show the quantity of a single item, or the sum total of all the merged items. For example, if a Three Cell Cyc with Circuits Report Detail is merged, you can show the Wattage of an individual circuit (Show totals of merged duplicates OFF) or the total Wattage of all three circuits (Show totals of merged duplicates ON).
- *Sort By*: Contains the settings for how the report will be sorted. The keys sort fixtures in alphabetical or numeric order. When fixtures have the same value in the first sort key, the report is then sorted by the values of the second sort key, and so on for the third sort key. For example, the standard Instrument Schedule sorts fixtures by position as Key 1, and unit number as Key 2. When a column is selected as a key, the visible option will be checked and greyed out. Columns that are sort keys must be visible.
- *Filters*: Contains settings for the column that you selected from the Columns list. You can use filters to extract specific information from reports. A total of three filters can be applied to any given report. For example, you can create a report that displays all the fixtures that have notes. To do so, select the Notes column and the *Visible* check box, and then select the filter *Not Empty*. You will also have to make Position, Unit, and Channel visible so that you know which fixtures have these notes.

To create a new report

1. Click the Reports shortcut tab.
2. Right-click in the reports shortcut area, and then choose **New Report**.
3. Type the name of your new report.
4. Select the Report Type (Fixtures, Truss, etc.).
5. Select an existing report from which you would like to copy the columns, formatting, filters, etc.
6. Click *OK*.

Redesigning Headings

You can customize the fonts, styles, and layouts used in the headings and text of your reports. These settings can be applied to the headings of all reports in your document as desired.

To change the heading format

As of Release 26, report headings can be directly modified on the current report. You may add, remove, and change the formatting of cells within the report with a limited degree of freedom. For more information about the functionality of the spreadsheet, refer to Worksheets on page “Worksheets” on page 464.

To apply the current formatting to other reports

Once the editing has been completed, you can apply the look and feel of your current report by completing the following steps:

1. Select **Report > Apply Current Report Template**.
2. In the new dialog, select the reports to which you want to apply the current format.
3. Click *OK*.

To export report formatting

Alternatively, you can save the current report formatting to a file, as follows:

1. Select **File > Export > Report Template**.
2. Select the file format that you would like to export to.
3. Click *OK*.

   Note: For optimal compatibility with other WYG files, it is recommended that you export to the WYSIWYG Spreadsheet format (*.wss).

To import report formatting

You can import a report format into WYSIWYG as follows:

1. Select **File > Import Report Template**.
2. Select the report file.
3. Click *OK*.
4. From the new dialog, select the reports to which you want to apply the template.
5. Click *OK*.

Column widths

You can change the column widths in the report. A double-headed arrow appears when you float the cursor between the column separations at the top of the report. Double-click the column headings to have the column resize to fit (auto-fit) the contents.

New Plots

Introduction

New Plots views were introduced so that you could more effectively manipulate a drawing developed in CAD mode to create a lighting plot or “schematic” for printing.

In the New Plots view you can:

- manipulate all fixtures and hang structures in a plot without affecting the objects in the original CAD drawing/model.
- notate your plot using objects such as circles, lines, arrows, and text labels.
- rotate the position of vertical hang structures to better illustrate mounted fixtures.
- move, rotate, or manipulate the layout of fixture attributes with click-and-drag.

Working in the New Plots view

Working in a New Plots view is similar in some respects to working with a drawing in a CAD wireframe view. You can use your keyboard and mouse to navigate around the plot easily and quickly. A command line interface is available to help you when entering coordinates.

IMPORTANT: Changes made to a drawing in CAD mode are not automatically reflected in the New Plots view. You must update plots manually to see the changes. For details, see “Updating plots” on page 419.

Keyboard and mouse control

Navigating in a New Plots view is similar to navigating in CAD mode. The arrow keys are available to help you move in the direction that is noted on the key. As well, scroll bars along the side and bottom of the window will move you horizontally and vertically along the window.

For a complete list of the navigation tools that are available, refer to “Keyboard and mouse control” on page 71.

Command line

The ComEdit toolbar is available to help you enter coordinates when placing objects such as lines in a New Plots view. For more information on command line, refer to “Command line” on page 103.

Creating a new plot

Plots are 2D schematics of your CAD model for printout. You can generate as many plots as you like. Once generated, the “view type” of the New Plot or schematic is set and cannot be changed. There are five plot types available.

Plot Type	Description
Plan Plot	Generates the schematic from above looking down.

Plot Type	Description
Left Section Plot	Generates the schematic looking from the left side through the venue.
Right Section Plot	Generates the schematic looking from the right side through the venue.
Front Elevation Plot	Generates the schematic looking from the front side through the venue.
Back Elevation Plot	Generates the schematic looking from the back side through the venue.

To create a new plot type

1. Click the *New Plots* shortcut tab.
2. Right-click in the *New Plots* shortcut area and choose the type of plot that you want to create.
3. Type the name of the plot.
4. Click *OK*.

   Result: The type of plot that you chose is created and a shortcut to your new plot is added to the New Plots shortcut bar. For information on working with shortcuts, refer to “Working with shortcuts” on page 56.

   Note: The plot is created with a specific set of default properties, but you can modify these settings as required. For more information on the settings, refer to “Modifying the properties of a plot” below.

Updating plots

Changes made to a drawing in CAD mode are not automatically reflected in the New Plots view. You must update plots manually.

To update a plot

1. Select the plot that you want to update.
2. From the **Options** menu, choose **Update New Plot**.

   Note: If the New Plots view already reflects the current CAD drawing, the option is disabled.

Modifying the properties of a plot

You can modify the border thickness, paper size, or scaling of a plot.

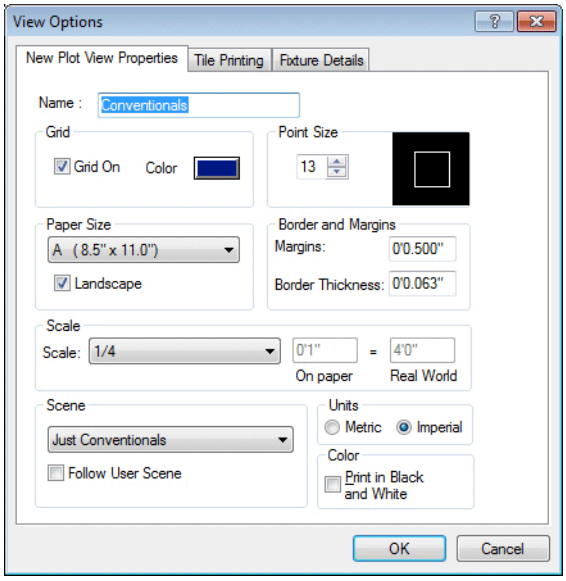
To modify plot properties

1 Select the plot for which you want to modify the settings.

2 From the **Options** menu, choose **View Options**.

Tip: You can also use the *View Options* tool on the *Standard* toolbar.

Result: The *View Options* dialog box appears.



New Plot View Properties tab

Options on the *New Plot View Properties* tab affect the appearance of the plot.

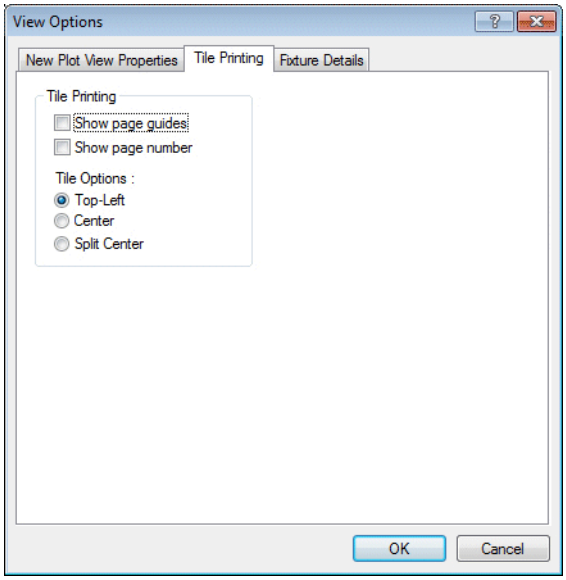
- *Name*: The name of the plot.
- *Grid On*: Select this check box if you want gridlines to appear on the plot view. Gridlines are for reference only; they assist with the layout of plot items on the page. Gridlines do not appear in the printed output.
- *Point Size*: The size at which points are printed.
- *Paper Size*: Choose the desired paper size from the drop-down menu. If you do not find the size you want, choose *Custom*, and then specify the width and height as desired. The default values are 24.0” X 36.0” (imperial measurement) or A1 (metric measurement).
- *Landscape*: Select this check box if you want the page to be set up in landscape format.
- *Margins*: The margins for the plot. The default values are 1/2” (imperial measurement) or 1cm (metric measurement).
- *Border Thickness*: The thickness of the plot’s border. The border is drawn at the margins defined above. The default values are 1/16” (imperial measurement) or 1mm (metric measurement).
- *Scale*: The scale at which the plot is drawn. If the scale you want to use for printing is not displayed, select *Custom*. The default values are 1/4”=1’0” (imperial measurement) or 1:50 (metric measurement).
- *On paper:* The scale ratio that you want to use on a paper copy of the plot view.
- *Real World*: The “real” measurement for the scale ratio that you specify in the *On paper* box. For example, you may want to use a scaling of one inch in a paper copy of your plot. This one-inch measurement translates to a “realistic” measurement (for example, 3 feet).
- *Follow User Scene*: Select this check box to use the scene that is displayed in the drop-down list on the *Scene* toolbar. Click to clear the check box, and then select

the desired scene from the *Scene* drop-down list. For more information on scenes, refer to “Scenes” on page 223.

- *Units*: The units of measurement for the plot. The coordinates at the bottom of the plot change according to the selection. This setting does not affect the printed output.
- *Print in Black and White*: Select this check box if you want the plot to be displayed and printed in black and white.

Tile Printing tab

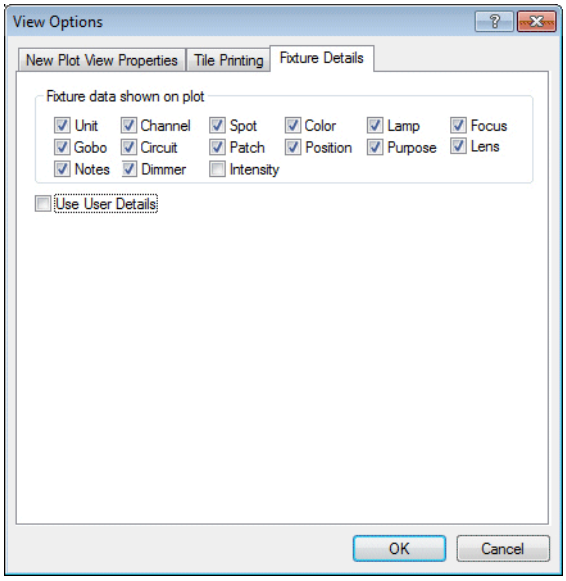
Options on the *Tile Printing* tab affect the printout of the plot.



- *Show page guides*: Select this check box if you want the printable outlines to be displayed on each sheet of paper that comprises the plot. The solid outline is the chosen printer’s paper size and orientation.
- *Show page number:* Select this check box if you want the page numbers of the plot to be displayed in the center of each page. The page numbers are for reference only and do not appear in the printed output.
- *Tile options*: Click the appropriate option button to enable one of the following:
  - *Top-Left* Click to place the top-left corner of the plot in the top-left corner of the first page, and then arrange the remaining pages accordingly.
  - *Center* Click to place the center of the plot at the center of a page, and then arrange the remaining pages accordingly.
  - *Split Center* Click to place the center of the plot at the corner of four pages, and then arrange the remaining pages accordingly.

Fixture Details tab

Options on the *Fixture Details* tab affect the fixture data that is displayed on the plot.



- *Use User Details*: Click to clear this check box if you do not want the currently selected fixture data displayed on the plot. The default fixture data is defined on the *Show Details* tab in *User Options*. For more information on this option, refer to “Show Details tab” on page 76.

Moving the page

You can move the page around to capture the parts of the schematic that you want plotted or printed.

To move the page

1. Select the plot that you want to adjust.
2. Right-click and choose **Move Page**.

   Result: A hand similar to the Pan tool is displayed.
3. Hold down the left mouse button and drag to move the page until the area of the schematic that you want on the page is contained within.
4. When you are finished, right-click and select **Finish Move Page**.

Objects in New Plots

Objects that are contained in the drawing in CAD mode are transposed to the New Plot representation. This representation is a 2D presentation of the drawing, and as such, certain objects are displayed differently than they are in the 3D CAD mode drawing.

A specific number of objects are used to represent the objects in the CAD drawing. These objects are listed in the following table.

Object in CAD mode	Resulting object in New Plots
Fixtures	Representative symbols
Accessories	Representative symbols
Riser	Riser

Object in CAD mode	Resulting object in New Plots
Text	Text
Arc	Arc
Ellipse	Ellipse
Circle	Circle

Notes:

- The only objects that you can edit in New Plots mode are fixtures and hang structures; you cannot select or modify any other object in your model in New Plots mode.
- Beams are not represented in the New Plots view.
- Some objects may not be visible if they are on a layer that is not in the current scene.

Relationship between objects in CAD mode and New Plots

Fixtures and hang structures are transposed to a New Plots view from CAD mode and can be manipulated independently of their counterparts in CAD. This means that in a New Plots view, they can be deleted, copied, moved, and have their properties modified without affecting the original object in CAD mode. For example, moving a pipe does not cause the same pipe to also move in CAD mode. You can easily create your lighting plot in a New Plots view independently of the drawing or “model” in CAD mode.

In contrast, changes made to the original “model” in CAD mode *are* reflected in all New Plots views. Objects that are deleted in CAD mode are also deleted from all New Plots views. Similarly, objects added in CAD mode are also added to all New Plots views, provided the layer they are on is visible. Note, however, that in order for new objects to appear, you must first update the New Plot by right-clicking on it and selecting *Update New Plot*. The same logic applies to objects that are moved in CAD mode: moving an object causes the object to also move in all New Plots views as long as the relationship between the object in CAD mode and New Plots is intact.

To achieve a relationship between fixtures and hang structures in CAD mode and fixtures and hang structures in New Plots views, various types of links are available. These links track the position and properties of the original CAD object in New Plots, and are referred to as the positional and properties links, respectively. For more information on these links and how they work, refer to “Links” on page 423.

Relationship between objects in Data mode and New Plots

You cannot modify data, such as fixture attributes, directly in a New Plots view; instead, you must edit this data in Data mode. Data that is modified in the spreadsheet view in Data mode is also updated in CAD mode and in all New Plots views.

Links

There are three types of links in the New Plots view: positional, properties, and attributes.

The position link tracks the movement of fixtures and hang structures that are brought in from the original CAD drawing. By default, this link is in place until one of the fixtures and hang structures from the original CAD drawing is moved relative to its origin.

The properties link tracks the changes to properties for fixtures and hang structures that are brought in from the original CAD drawing. These properties include the layer line color and line weight of an object. By default, this link is in place until you modify one or more of these properties.

The attribute link tracks the changes to the attribute layout of fixtures. By default, this link is in place until you move, rotate, or otherwise manipulate the layout of a fixture attribute. For more information on attribute layout, refer to “Attribute layout” on page 426.

When a link to an object is broken, all subsequent changes that are made to that object in CAD mode will not be reflected in a New Plots view. You must reestablish the relationship or link to ensure that changes made in CAD or Data mode are reflected in a New Plots view.

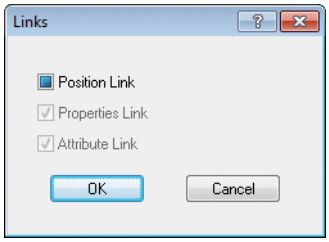
Reestablishing the link returns the object to its original state before the link was broken. For example, reestablishing the link of a fixture that has been moved off a pipe moves the fixture back to its original position on the pipe.

To reestablish links

1 Select the object for which you want to reestablish a link.

2 From the **Edit** menu, choose **Restore Links**.

Result: The *Links* dialog box is displayed.



Note: The *Position Link* check box is deselected. This means that an action occurred that broke that link.

3 To reestablish the link, select the *Position Link* check box.

4 Click *OK*.

Result: The positional link is reestablished.

Vertical hang structures

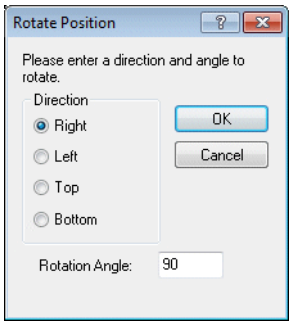
In the New Plots view, non-horizontal hang structures, such as booms and ladders can be rotated around their base to show all mounted fixtures. The rotation can be top, bottom, left, or right.

To rotate the position of a vertical hang structure

1 Select the hang structures that you wish to rotate.

2 Right-click and choose **Rotate Position**.

Result: The *Rotate Position* dialog box is displayed.



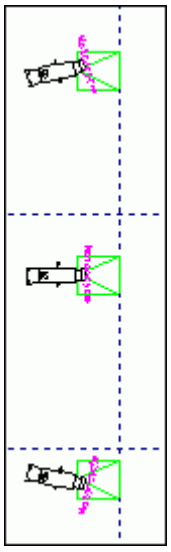
3 Select the direction in which you want to rotate the hang structure.

4 In the *Rotation Angle* box, type the angle by which you want to rotate the hang structure.

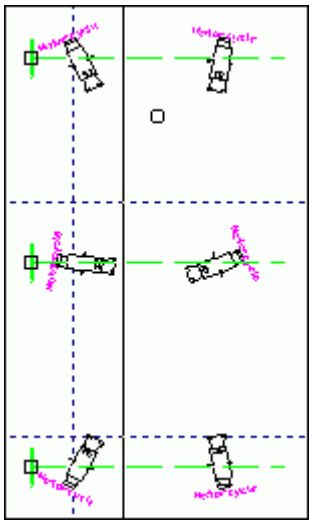
5 Click *OK*.

Result: The hang structure is rotated on the plot.

As an example, you may have three booms on your plot that you want to rotate.



After you rotate the hang structures 90 degrees to the right, the following is displayed.



Attribute layout

You can easily modify the layout of fixture attributes in a New Plots view by simply moving, rotating, or otherwise manipulating the appearance of the attribute. In a New Plots view, you can only manipulate the layout; you cannot modify the data contained within the attribute. To modify the data, you will need to go to CAD or Data mode and make the modifications.

Fixture attribute values that are changed in CAD or Data mode are reflected in all New Plots views.

The attributes that are displayed in a New Plots view are determined by the options you set on the *Fixture Details* tab. For more information, refer to “Fixture Details tab” on page 422.

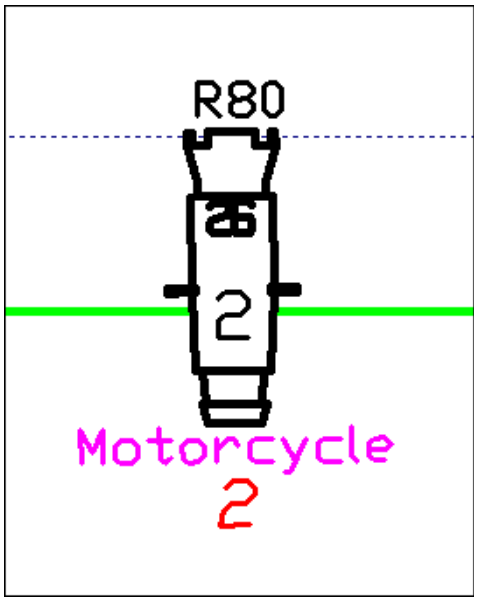
To manipulate the attribute layout for an individual symbol

1 Select the attribute whose layout you want to manipulate.

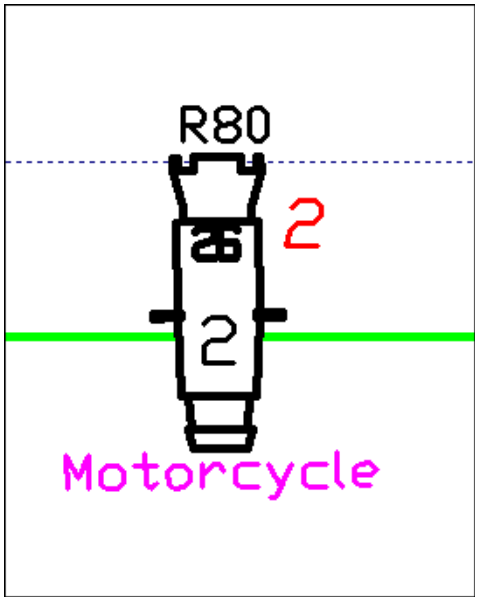
2 Drag the attribute to the desired location on the page, and then release the mouse button.

Note: This action breaks the attribute link.

As an example, you may wish to move the channel attribute (the red “2”) shown on the plot below.



After you select and drag the attribute to the desired location, the following is displayed.



Drawing objects

You can add objects to the plot for notation. Objects that you can draw include lines, arcs, circles, and text labels. You can also insert symbols from the Library Browser.

Drawing circles

Drawing circles in a New Plots view is similar to drawing circles in CAD mode. You can access the Circle tool from the **Draw** menu or from the *New Plot Options* toolbar  . For more information on drawing circles, refer to “Drawing circles” on page 146.

Note: When you add objects such as circles to your plot, they are not added to your drawing in CAD mode.

Drawing arcs

Drawing arcs in a New Plots view is similar to drawing arcs in CAD mode. You can access the Arc tool from the **Draw** menu or from the *New Plot Options* toolbar  . For more information, refer to “Drawing arcs” on page 146.

After you have drawn an arc, you can choose to add arrows to one or both ends of the arc. The default is none. For more information on adding arrows, refer to page 434.

Drawing lines

Drawing lines in a New Plots view is similar to drawing lines in CAD mode. For more information, refer to “Drawing lines” on page 144.

You can also use the appropriate line tool from the **Draw** menu or the *New Plot Options* toolbar. The available line tools are as follows:

-  Solid
-  Dot
-  Center
-  Dash

You can create a double line from a single line by modifying the properties of a line. For more information on creating double lines, refer to page 434.

After you have drawn a line, you can choose to add arrows to one or both ends of the line. The default is none. For more information on adding arrows, refer to page 434.

Note: When you add objects such as lines to your plot, they are not added to your drawing in CAD mode.

Drawing text

Drawing text in a New Plots view is similar to drawing text in CAD mode. You can access the Text tool from the **Draw** menu or on the *New Plot Options* toolbar  . For more information, refer to “Drawing text labels” on page 148.

Notes:

- When you add an object such as text to your plot, it is not added to your drawing in CAD mode.
- You can select any font that is installed on your computer to use in all text labels in New Plots. For details, see “To change the font used in New Plots” on page 428.

To change the font used in New Plots

You can specify the font that you would like to appear globally in all *new* text labels that you create in New Plots.

Note: When you change the font globally, it does not affect *existing* labels in your file, only new ones that you draw. You cannot change the font of existing labels in New Plots view.

1. In New Plots view, click **Options > User Options**.
2. Click the *Font Styles* tab.
3. Under New Plots Text Labels, from the *Font* drop-down list, select the font that you want to appear in all new text labels.
4. Choose whether you want the letters to appear bold, underlined, or in italics.
5. Click *OK*.

Inserting symbols

You can insert symbols that are not associated with any object in CAD mode. This allows you to represent the fixtures when you draw a key or want to show the position of a boom in a plan view, for example. You can access the Symbol tool from the **Draw** menu or from the *New Plot Options* toolbar  .

To learn how to insert symbols, see “To insert objects from the library” on page 135. In the New Plots view, the Library Browser contains only the Symbol library.

Editing objects

A number of tools are available in the New Plots view to help you manipulate objects.

Note: Orthographic mode is available in New Plots from the **Edit** menu and also from the *New Plots Options* toolbar. When ortho mode is active, you can manipulate objects either vertically or horizontally. For more information on this mode, refer to “Ortho” on page 105.

Selecting objects

Objects are transposed from CAD mode and can, therefore, be selected independently of that mode (for example, moving a pipe in New Plots will not cause it to also move in CAD mode).

Fixture symbols are grouped with the hang structure to which they are mounted. Selecting and moving the hang structure will also move the mounted fixture symbols. Similarly, accessories are grouped with the fixture symbols to which they are attached and will also move with the fixture symbols. You can also select an individual fixture symbol separately from the hang structure if you want to move or modify it independently.

The methods for selecting an object are similar to those for selecting an object in CAD mode. For more information on these methods, refer to “Selecting” on page 185.

Copy

When you copy and paste an object in a New Plots view, a second representation of that same object is created. If you subsequently update the data for this object in either CAD or Data mode (for example, gel color or channel number), all representations of that object in all New Plots views are updated.

Copied objects are not added to the drawing in CAD mode nor to the spreadsheet in Data mode.

You can copy and paste objects between New Plots views.

To copy an object

1. Select the object to be copied.
2. From the **Edit** menu, choose **Copy**.
3. From the **Edit** menu, choose **Paste**.
4. Click in the plot where you want to place the object.

Cut

When you cut an object in a New Plots view it is not removed from CAD mode. However, cutting an object in CAD mode removes it from all New Plots views.

Cutting an object adds it to the clipboard. You can subsequently paste the object into a New Plots view.

You can cut and paste objects between New Plots views.

To cut an object

1 Select the object to be cut.

2 From the **Edit** menu, choose **Cut**.

3 From the **Edit** menu, choose **Paste**.

4 Click in the plot where you want to place the object.

Delete

Deleting an object in a New Plots view does not delete the original object in CAD mode. However, deleting an object in CAD mode does delete the representation of that object in all New Plots views.

To delete an object

1 Select the object to be deleted.

2 From the **Edit** menu, choose **Delete**.

Result: The object is removed from all New Plots views.

Note: If you accidentally deleted an object, you can recover it by choosing **Undo Delete** from the **Edit** menu.

Move

The Move command in the New Plots view supports both absolute and relative moves in a manner similar to CAD mode. For more information on this command, refer to “Moving” on page 191.

When you move a hang structure, you also move all its attached fixtures and accessories. You also break the positional link between the pipe in CAD mode and the pipe in the New Plots view. All fixtures and accessories that are attached to that pipe still maintain their link to the original objects in CAD mode, even though they have also moved.

When you move a fixture off of a hang structure, you fix its location in relation to the hang structure. For example, if you move a fixture off a pipe, and then move the pipe 10 feet, the fixture also moves with the pipe. This action breaks the positional link of the fixture to the hang structure.

Note: You can “nudge” objects up, down, left, or right a certain distance by using specific key combinations. Use the ALT+arrow keys to “nudge” an object a specific distance; use the ALT+SHIFT+arrow keys to “nudge” an object a tenth of that distance.

Align

You can use the Align tools to quickly and precisely align a series of objects that you have drawn or inserted from the Library into your drawing. For example, if you have manually placed some chairs in your drawing, you can select them all and align them in one of six ways.

Notes: You cannot use the Align tool to align fixtures.

You can align the selected objects from the:

- top

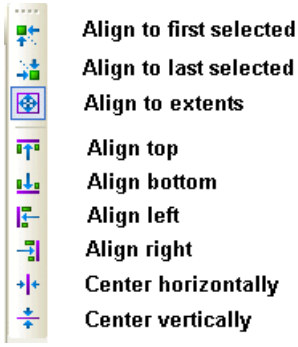
- bottom
- left
- right
- center (horizontally)
- center (vertically)

Before choosing any of these options, you must select one of following three options:

- **To First Selected**: Objects selected after the first object was selected will align (Left, Right, Centered Horizontally, etc.) to that first object.
- **To Last Selected**: Objects selected before the last object was selected will align (Left, Right, Centered Horizontally, etc.) to that last object.
- **To Extents**: All selected objects (regardless of selection order) will align to the left-most, right-most, bottom-most, etc. point of the left-most, right-most, bottom-most, etc. object, respectively.

To align objects

1 In the toolbar area, ensure that the *New Plot Align* toolbar is showing.



2 Select the objects that you want to align by clicking and dragging a selection box around them, or selecting them individually.

3 On the *New Plot Align* toolbar, choose your preferred alignment option.

- Align to First Selected
- Align to Last Selected
- Align to Extents

Note: Before you align the objects, it is recommended you first verify that they will not overlap each other in their new positions.

4 On the *New Plot Align* toolbar, select how you would like to align the selected objects:

- Top
- Bottom
- Left
- Right
- Center Horizontally
- Center Vertically:

Rotate

The Rotate command in a New Plots view works in a similar manner as it does in CAD mode. For more information on this command, refer to “Rotating” on page 193.

Note: Rotating an object breaks its positional link.

Fillet

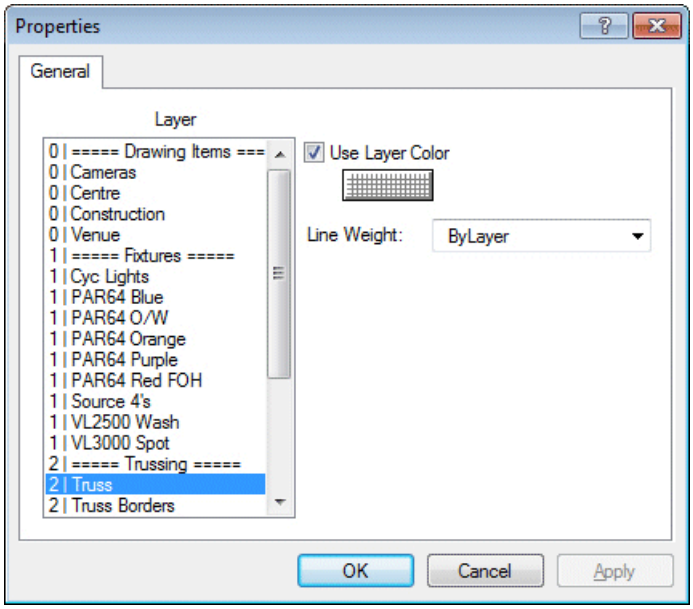
The Fillet command in a New Plots view works in a similar manner as it does in CAD mode. For more information on this command, refer to “Fillet” on page 234.

Objects that are brought into New Plots from CAD mode

There are general properties for each fixture or hang structure that is brought into a New Plots view from CAD mode. You can modify these general properties as required.

General tab

Options on the *General* tab affect the selected object’s color and line weight properties.



- *Layer*: A list of the layers in your document is displayed. Click on a list item to change the layer on which the object will be drawn.
- *Use Layer Color*: Select this check box to set the object’s color for wireframe views to be the same as the properties of the layer on which the object resides. For more information, refer to “Layer properties” on page 220.

  Click to clear the check box, and then use the color button to set a specific color for the object.
- *Line Weight*: From the drop-down list, choose the specific line weight for the selected object. The applied line weight is visible in all Wireframe views in all modes. For objects only, you can also choose the default setting, *ByLayer*, which sets the object’s line weight to be the same as the properties of the layer on which the object resides. The value *Default* means that the document's line weight setting from the Show Options > Object Settings window is applied to the object or layer. For more information, refer to “Layer properties” on page 220.

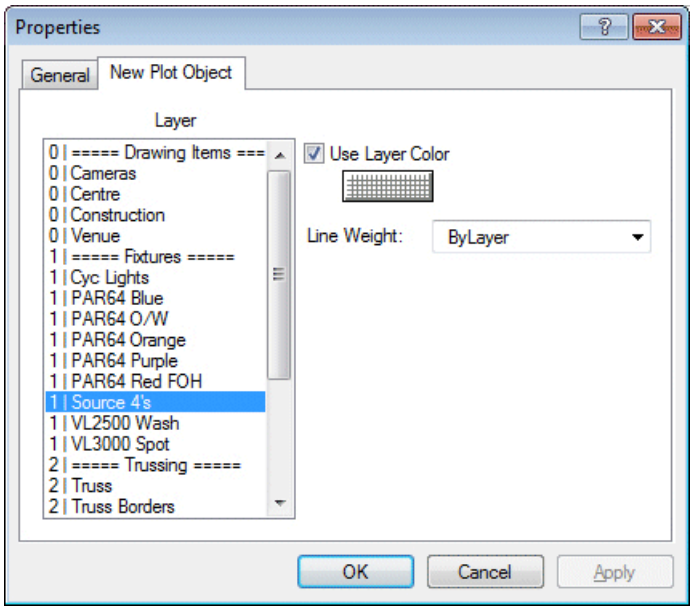
Objects that are drawn in New Plots

Objects such as lines, arcs, symbols, circles, and text that are drawn in a New Plots view have general properties that are inherited from the layer on which they are drawn. Along with the general properties, a tab appears in the properties dialog box for each type of object that is selected. When you select multiple objects of different types, tabs appear for each type of object selected.

For more information, refer to “Drawing objects” on page 138.

New Plot Object tab

Options on the *New Plot Object* tab affect the selected object's color and line weight properties.

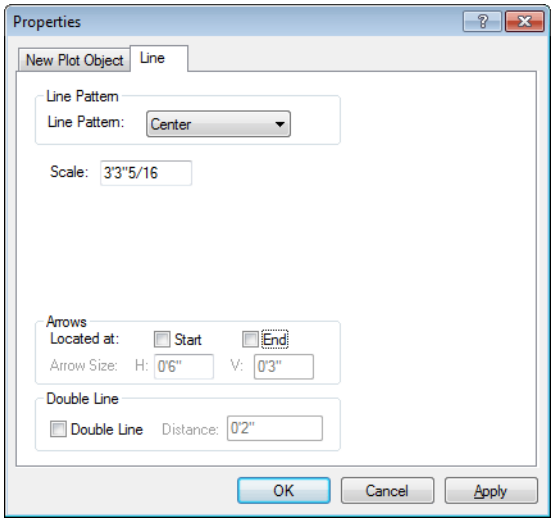


For a detailed description of the options available on this dialog box, refer to the "General tab" on page 432.

Object-specific properties

Similar to objects in CAD mode, objects that are drawn in a New Plots view have properties that are specific to that object.

- **Text label** - For a description of text label properties, refer to the "Text label tab" on page 207.
- **Symbol** - For a description of symbol properties, refer to "Symbol properties" on page 314.
- **Line** - Options on the *Line* tab affect how the line is drawn.

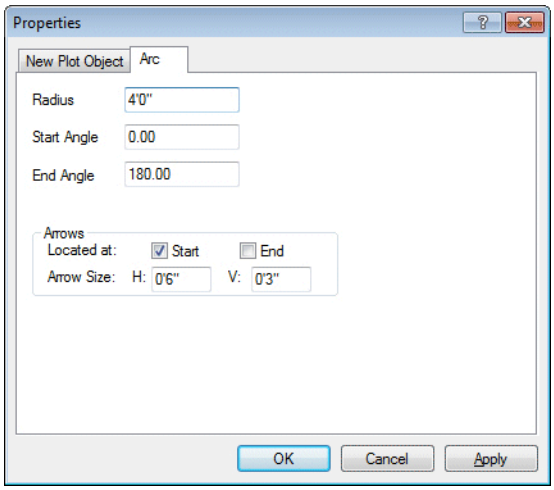


- *Line Pattern*: Choose a pattern for the selected line. Refer to "Drawing lines" on page 144 for an illustration of each type.

- *Scale*: Type a value in this box to change the length and spacing of dots and dashes for the selected line. This value is applicable to center, dashed, or dotted lines only.
- *Arrows*: Select where you would like to add arrows on the line. You can specify a width and length for the arrows by entering values in the *H* and *V* boxes, respectively.
- *Double Line*: Select this check box to change the selected line to a double line. Indicate how far apart you want the double lines to be by entering a value in the *Distance* box.

Circle - The properties are the same as when the object was originally drawn. For more information, refer to “Drawing circles” on page 146.

Arc - The radius and angle properties are the same as when the object was originally drawn. The only property that is different is the addition of arrows to either side of the selected arc.



- *Arrows*: Select where on the arc you would like to add arrows. You can specify a width and length for the arrows by entering values in the *H* and *V* boxes, respectively.

Inserting New Plots into layouts

Once you create your New Plot, you can insert it into the Layouts view. For example, you might want to insert it into a Layouts view so that you can add a legend, key, or report to your plot.

For more information on inserting a New Plot, see “To insert New Plot items onto the layout” on page 443.

Printing New Plots

While you can print directly from the New Plots tab, it is strongly recommended that you first insert your completed New Plot into a Layout and print it from there. Doing so enables you to take advantage of the Layout tools (and others), use title blocks, add images, and so on.

If you have a large plot you can “tile print” the plot across multiple smaller sheets of paper. To set up how you want the plot arranged on the sheets of paper, refer to the “Tile Printing tab” on page 421.

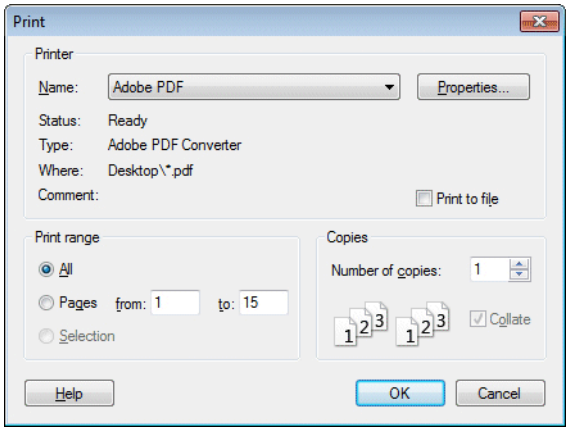
You can print your complete plot or specific pages of your plot based on the settings that you choose.

Note: Objects outside the border are not printed, even if they are within the margins of the page. To adjust margins, refer to “Modifying the properties of a plot” on page 419.

To print your plot

1 From the **File** menu, choose **Print**.

Result: The *Print* dialog box is displayed.



2 From the *Name* drop-down list, choose the printer (or plotter) that you want to use. WYSIWYG records the name that you choose for future reference and displays it as the default printer for plots.

3 To print all pages of a plot, select *All*.

4 To print select pages of a plot, select *Pages*, and then enter the range in the *from* and *to* boxes.

5 To print multiple copies of pages, select the value from the *Number of copies* list box.

6 Click *OK*.



Layouts

Introduction

The Layouts tab opens the layout design and storage area. Layouts are created by arranging various CAD, report, image, and other items on a defined paper size.

There are two default layouts included in the layouts shortcuts. The Micro Layouts, which is designed to print on an 8.5” x 11” piece of paper, and the Big Layouts, which is designed to be plotted on a 36” x 48” sheet of paper. You can use either of these layouts as is, modify them to suit your needs, or create new layouts from scratch.

To save time, you can use the Layout Groups feature to select multiple objects in your layouts and group them for insertion into other drawings. For details, see “Layout Groups” on page 443.

Creating a new layout

To create a new layout

1. Click the *Layouts* shortcut tab.
2. Right-click in the *Layouts* shortcut area, and then choose **New Layout**.
3. Type the name of the new layout.
4. Click *OK*.

   Result: An empty layout is created. A shortcut to your new layout is added to the Layouts shortcut bar.

   Note: By default, empty layouts are set up to use 8.5” x 11” sheets of paper.

Working with layouts

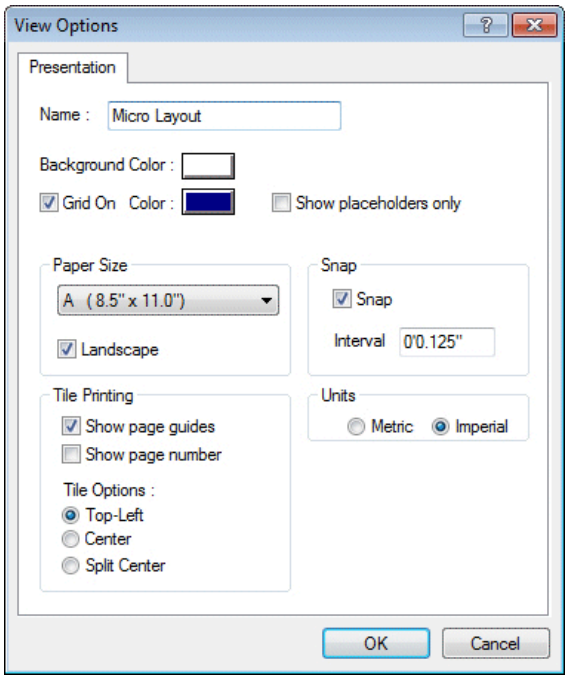
To modify layout settings

1 Select the layout for which you want to modify the settings.

2 From the **Options** menu, choose **View Options**.

Tip: You can also use the *View Options* tool on the *Standard* toolbar. 

Result: The *View Options* dialog box is displayed.



Presentation tab

Options on the *Presentation* tab affect the appearance and printout of the layout.

- *Name:* The name of the layout.
- *Background Color:* Sets the background color of the layout view. This does not affect your printed copy.
- *Grid On:* Select this check box if you want gridlines to appear on the layout. Gridlines are for reference only; they assist with the design of items on the page. Gridlines do not appear in the printed output.
- *Color:* Sets the color of the gridlines.
- *Show placeholders only:* Select this check box to display only the placeholders for the various items in the layout. If unchecked, both the placeholders and contents of the items in the layout are displayed. This is to facilitate the layout setup and does not affect the printout.
- *Snap*: Select this check box to snap the items in the layout to the gridlines.
- *Interval:* Specify the interval between the points on the grid.
- *Units*: The units of measurement for the layout. The coordinates at the bottom of the layout change according to the selection. This does not affect the printed output.
- *Paper Size*: Choose the desired paper size from the drop-down menu. If you do not find the size you want, choose *Custom*, and then specify the width and height as desired.
- *Landscape:* Select this check box if you want the page to be set up in landscape format.

- *Show page guides*: Select this check box if you want the printable outline to be displayed on each sheet of paper that comprises the layout. The solid outline is the chosen printer's paper size and orientation.
- *Show page number*: Select this check box if you want the page numbers of the layout to be displayed in the center of each page. The page numbers are for reference only and do not appear in the printed output.
- *Tile options*: Select one of the following option buttons:
  - *Top-Left* Places the top-left corner of the layout in the top-left corner of the first page, and then arranges the remaining pages accordingly.
  - *Center* Places the center of the layout at the center of a page, and then arranges the remaining pages accordingly.
  - *Split Center* Places the center of the layout at the corner of four pages, and then arranges the remaining pages accordingly.

Inserting items on layouts

Notes:

- To adjust the view of the Layout as you work on it, you can use your mouse wheel to zoom in or out and/or pan around with the middle mouse button.
- To copy and paste items in your Layout, select the item that you want to copy and click **Edit** > **Copy** (or press CTRL + C). Then, to paste the item, click **Edit** > **Paste** (or click CTRL + V) and click to place the pasted item.

Objects can be inserted from the following categories:

- basic shapes (lines, rectangles, ellipses, polygons, and text)
- images (bitmap and jpeg)
- CAD items
- data items
- report items
- worksheet items (includes the default Title Block)
- legend items
- key items
- new plot items

To insert a line onto the layout

1 From the **Insert** menu, choose **Line**.

Tips:

- You can also use the *Line* tool from the *Layout Pres Insert* toolbar. 
- Press SHIFT while drawing the line to draw it at a 45-degree angle, or along the X or Y axis.

2 Click on the page where you want to start drawing the line and, holding the mouse button down, drag it to the end point of the line. Release the mouse button when you have the desired shape.

3 Click anywhere in the drawing to finish placing the line.

Result: The line is placed on the layout.

To insert a rectangle onto the layout

1 From the **Insert** menu, choose **Rectangle**.

Tips:

- You can also use the *Rectangle* tool from the *Layout Pres Insert* toolbar. 
- Press SHIFT while drawing the outline of the rectangle to create a perfect square.

2 Click on the page at the starting point of the rectangle and, holding the mouse button down, drag out the shape of the rectangle. Release the mouse button when you have the desired shape.

3 Click anywhere in the drawing to finish placing the rectangle.

Result: The rectangle is placed on the layout.

To insert a rounded rectangle onto the layout

1 From the **Insert** menu, choose **Round Rectangle**.

Tips:

- You can also use the *Round Rectangle* tool from the *Layout Pres Insert* toolbar. 
- Press SHIFT while drawing the outline of the rectangle to create a perfect square.

2 Click on the page at the starting point of the rectangle and, holding the mouse button down, drag out the shape of the rectangle. Release the mouse button when you have the desired shape.

3 Click anywhere in the drawing to finish placing the rectangle.

Result: The round rectangle is placed on the layout.

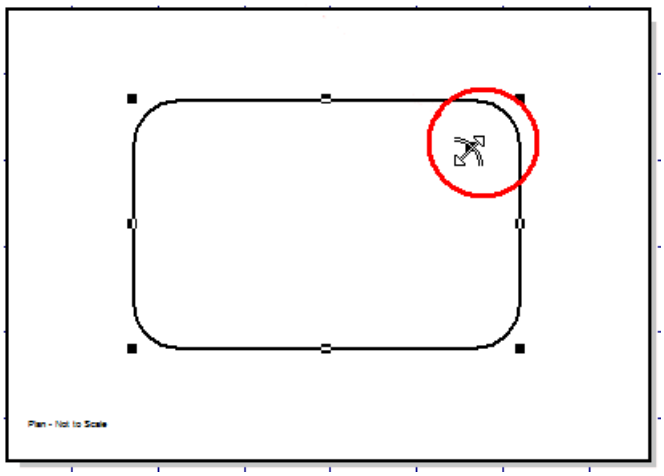
Note: You can only see the rounded corners of the rectangle if you zoom in on the drawing.

To adjust the corners of a rounded rectangle

You can adjust the roundness of the rectangle's corners by following these steps:

1 Select the rectangle.

2 Hover your cursor over the top-right corner until you see a curved arrow appear, as shown below:



3 Click and drag the corners of the rectangle inward to the desired amount of roundness.

To insert an ellipse onto the layout

1 From the **Insert** menu, choose **Ellipse**.

Tips:

- You can also use the *Ellipse* tool from the *Layout Pres Insert* toolbar. 
- Press SHIFT while drawing the ellipse to draw a perfect circle instead.

2 Click on the page at the starting point of the ellipse and, holding the mouse button down, drag out the shape of the ellipse. Release the mouse button when you have the desired shape.

3 Click anywhere in the drawing to finish placing the ellipse.

Result: The ellipse is placed on the layout.

To insert a polygon onto the layout

1 From the **Insert** menu, choose **Polygon**.

Tip: You can also use the *Polygon* tool from the *Layout Pres Insert* toolbar. 

2 Click on the page at the starting point of the polygon and, holding the mouse button down, drag out the shape of the polygon. Release the mouse button when you have the desired shape.

3 Click anywhere in the drawing to finish placing the polygon.

Result: The polygon is placed on the layout.

To insert text onto the layout

1 From the **Insert** menu, choose **Text**.

Tip: You can also use the *Text* tool from the *Layout Pres Insert* toolbar. 

2 Click on the page at the starting point of the text box and, holding the mouse button down, drag out the shape of the box. Release the mouse button when you have the desired shape.

3 Click anywhere in the drawing to finish placing the box.

Result: The text box is placed on the layout.

4 To type text in the new box, double-click the box, and then delete the default text.

5 Type your custom text.

Tip: Press SHIFT+ENTER to force a new line of text if Text Wrapping is enabled on the Text Toolbar.

6 Click anywhere in the drawing to finish adding the text.

Tip: You can use the Layout Pres Text toolbar to easily modify text; more options are available in the text object's Item Properties. You can also replace your layout text with variables by typing them directly. To do so, you must surround the variable with per cent symbols (%) when altering the text item's properties. For example, to add the Chief Lighting Designer variable, you would type %Chief Lighting Designer%. For more information on the variables available in Layouts, see "Smart Cells" on page 519.

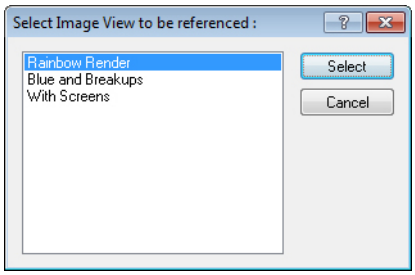
To insert image items onto the layout

1 From the **Insert** menu, choose **Image**.

Tip: You can also use the *Image* tool from the *Layout Pres Insert* toolbar. 

2 Click and drag a window (placeholder) on the page where you want the item to appear.

Result: The *Select Image View to be referenced* dialog box appears.



3 Select the image that you want to add to the layout, and then click *Select*.

Result: The image appears within the placeholder as drawn.

To insert CAD items onto the layout

1 From the **Insert** menu, choose **CAD**.

Tip: You can also use the *CAD item* tool from the *Layout Pres Insert* toolbar. 

2 Click and drag a window (placeholder) on the “page” where you want the item to appear.

Result: The CAD item, which is referenced from CAD mode, appears within the placeholder as drawn. By default, CAD items appear in plan view and at a scale of 1/16”=1’.

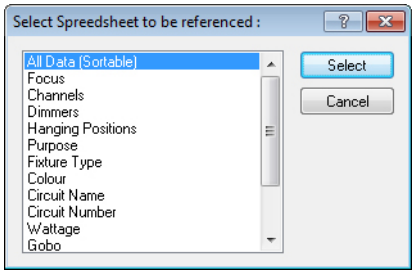
To insert data items onto the layout

1 From the **Insert** menu, choose **Data**.

Tip: You can also use the *Data* tool from the *Layout Pres Insert* toolbar. 

2 Click and drag a window (placeholder) on the “page” where you want the item to appear.

Result: The *Select Data View to be referenced* dialog box appears.



3 Select the data item that you want to add to the layout, and then click *OK*.

Result: The selected data sheet (referenced from Data mode) appears within the placeholder as drawn.

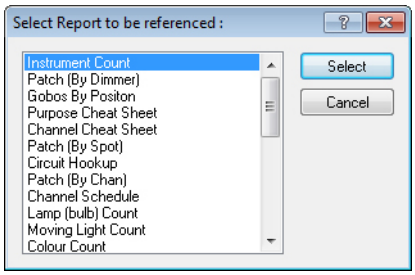
To insert report items onto the layout

1 From the **Insert** menu, choose **Report**.

Tip: You can also use the *Report* tool from the *Layout Pres Insert* toolbar. 

2 Click and drag a window (placeholder) on the "page" where you want the item to appear.

Result: The *Select Report to be referenced* dialog box appears.



3 Select the report item that you want to add to the layout, and then click *OK*.

Result: The selected report (referenced from Pres mode) appears within the placeholder as drawn.

Note: If the *Group By* option has been set for a report, the report cannot be added to a layout. Setting the *Group By* option creates page breaks within the report and layouts do not support reports with page breaks. For more information on this option, refer to "Report tab" on page 415.

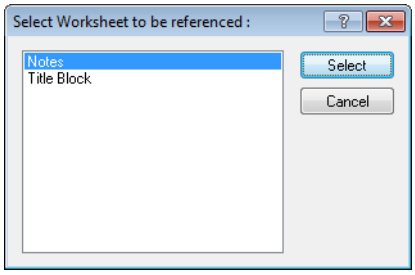
To insert worksheet items onto the layout

1 From the **Insert** menu, choose **Worksheet.**

Tip: You can also use the *Worksheet* tool from the *Layout Pres Insert* toolbar. 

2 Click and drag a window (placeholder) on the "page" where you want the item to appear.

Result: The *Select Worksheet to be referenced* dialog box appears.



3 Select the worksheet that you want to add to the layout, and then click *OK*.

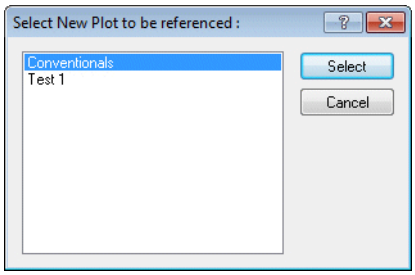
Result: The selected worksheet (referenced from Pres mode) appears within the placeholder as drawn.

To insert New Plot items onto the layout

1 From the **Insert** menu, choose **New Plot.**

2 Click and drag a window (placeholder) on the "page" where you want the item to appear.

Result: The *Select New Plot to be referenced* dialog box appears.



3 Select the New Plot item that you want to add to the layout, and then click *OK*.

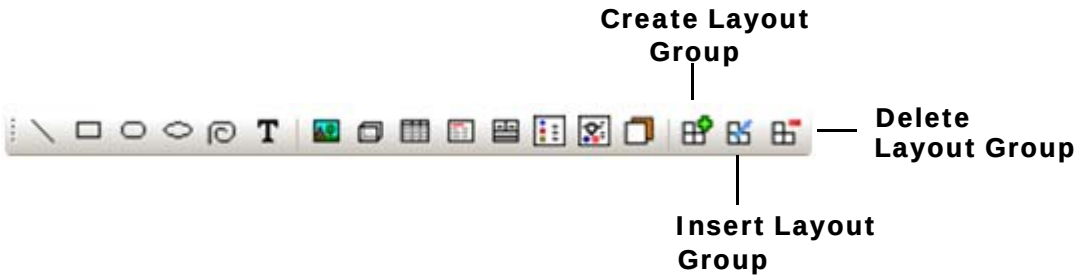
Result: The selected New Plot item (referenced from Pres mode) appears within the placeholder as drawn. To adjust the contents that are displayed within the placeholder, refer to "Adjusting the contents" on page 449 for further details.

Layout Groups

To save time, you can use the Layout Groups feature to select multiple objects in your layouts and group them for insertion into other drawings.

Layout Groups icons

The Layout Pres Insert toolbar includes the three icons for creating, inserting and deleting Layout Groups. You can also use the menu commands, as detailed in the procedures below.



To create a new Layout Group

1 In your Layout, select the items that you want to include in the Layout Group. For example, you can select Title Blocks, Plot Notes, Custom Legends and Keys, Disclaimers, and so on.

2 Right-click and select **Create Layout Group**.

3 Type a name for your new Layout Group.

4 Click *OK*.

To insert a Layout Group in your layouts

Once you have created a Layout Group, you can insert it into other drawings as follows:

1 In your Layout, right-click and select **Insert Layout Group**.

2 In the window that appears, select the name of the Layout Group that you want to insert, and then click *OK*.

3 Navigate to the location where you want to insert the Layout Group and click to place it.

Tip: If you do not select any items in the Layout, saving a Layout Group will prompt you to save the entire Layout as a Layout Group. This includes Layout view options, such as Paper Size and Tile Printing. To insert the entire Layout, right-click in the Shortcut Bar (Layouts page), and select **New Layout Using Layout Group**.

Tip: If no items in a Layout are selected, saving a Layout Group will prompt you to save the entire Layout as a Layout Group. This includes Layout view options, such as Paper Size and Tile Printing. To insert the entire Layout, right-click in the Layouts shortcut bar and select **New Layout Using Layout Group**. Type a name for the new Layout, and then click *OK* to create it. The new Layout appears on the Layouts shortcut bar.

To delete a Layout Group from the database

Once you have created a Layout Group, you can delete it from the database, which means it will no longer be saved with your WYSIWYG file and you will no longer be able to insert it in subsequent drawings.

1 On the Layouts tab, on the Layout Pres Insert toolbar, click the Delete Layout Group icon .

2 In the window that appears, select the name of the Layout Group that you want to delete, and then click the *Delete* checkbox.

3 Click *OK* to delete the Layout Group.

Content and Item properties

As with all objects in WYSIWYG, the objects that you have placed on your page have unique properties. There are two types of properties:

- Content Properties
- Item Properties

Content Properties

Content Properties affect the content of the presentation item. When you open these properties, you open a similar or identical dialog box to the *View Options* dialog box that appears when you are working on the original item in CAD, Data, or Pres mode. In this dialog, you can modify the information to be displayed and control how it is displayed.

To modify content properties

1 Select the presentation item that you want to modify.

2 From the **Edit** menu, choose **Content Properties**.

Tip: You can also click the *Content Properties* tool on the *Edit* toolbar.

Result: The *View Options* dialog box is displayed.

Note: Content properties for CAD items are discussed in “CAD items” on page 445.

Item Properties

Item Properties affect the placeholders and the scaling of the contents. Options here include the outline’s weight and color, and various printing options for the contents, such as scale.

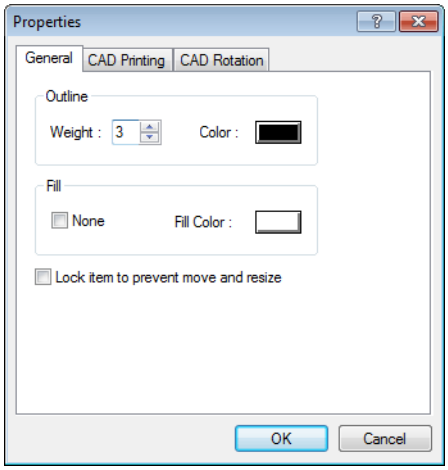
To modify item properties

1 Select the presentation item you want to modify.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Edit* toolbar. 

Result: The *Properties* dialog box is displayed.



Note: The first page of the properties dialog is identical regardless of the type of presentation item. Options here affect the outline and fill color of the placeholder.

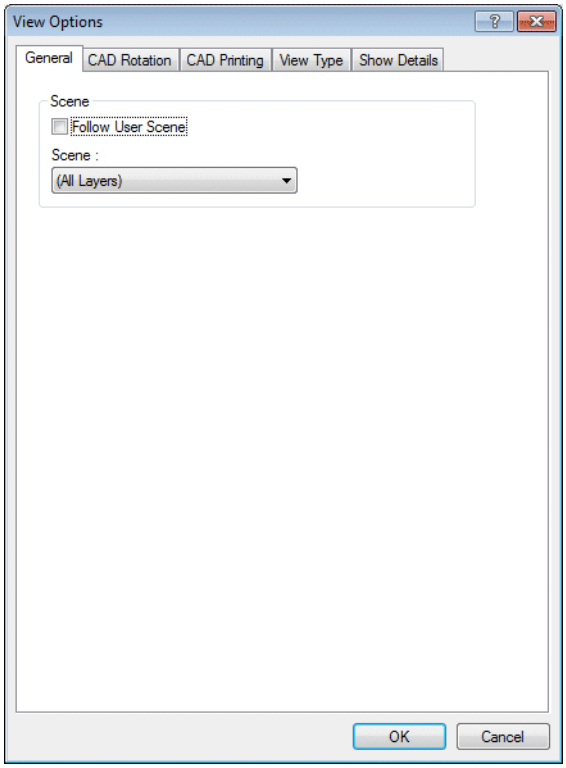
CAD items

You can modify or edit CAD items to meet printing requirements. Their content properties are detailed below.

To modify content properties of CAD items

1 Select the CAD item, right-click, and then choose **Content Properties** or click the appropriate tool on the *Layout Edit* toolbar.

Result: The *View Options* dialog box is displayed.



2 Modify options as desired.

3 When finished, click *OK*.

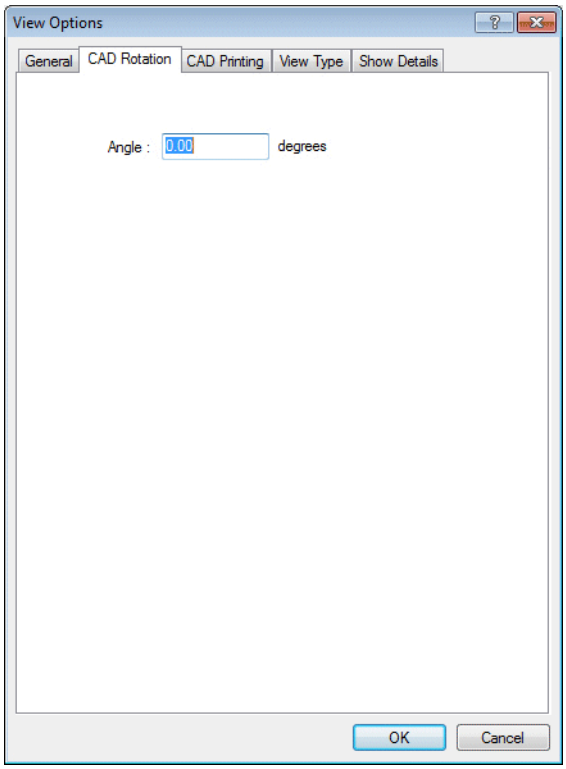
General tab

Options on the *General* tab affect the scene that is displayed in the CAD item. Only fixtures and objects on the active layers are displayed in the layout.

- *Follow User Scene*: Select this check box to use the scene that is displayed in the drop-down list on the Scene toolbar. Click to clear the check box, and then select the desired scene from the Scene drop-down list. For more information on scenes, refer to “Scenes” on page 223.

CAD Rotation tab

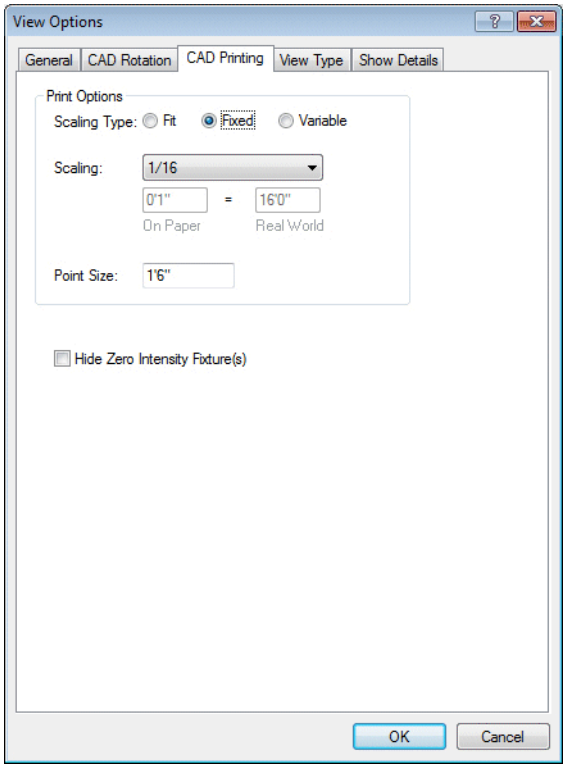
Options on the *CAD Rotation* tab affect the angle at which the CAD item is printed.



- *Angle:* Type an angle value in degrees if you want the drawing in the CAD item to be printed on an angle. This feature is helpful for printing booms.

CAD Printing tab

Options on the *CAD Printing* tab affect the scale for the drawing.



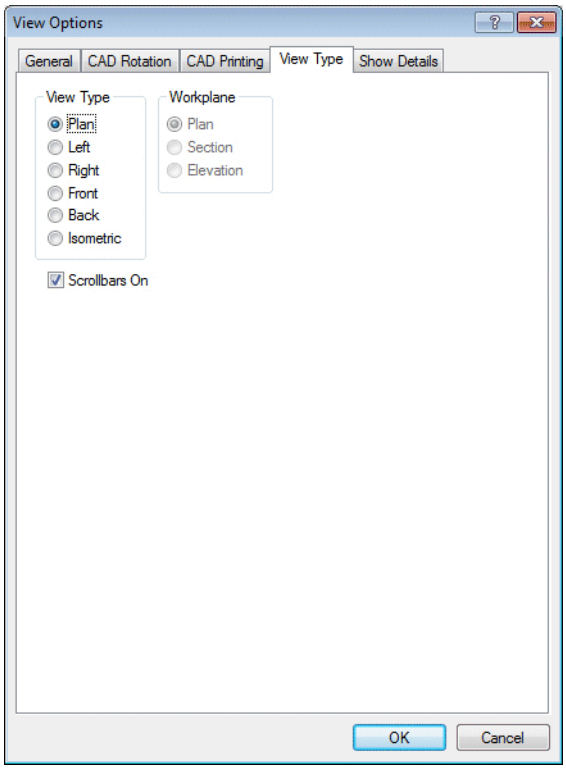
- *Fit*: If you select this option, the entire CAD drawing is scaled to fit inside the placeholder.
- *Fixed:* If you select this option, the CAD item is printed at the specified scale. To determine which portion of the CAD item is displayed inside the placeholder,

choose **Open Contents**. Inside this view, you can move around a white rectangle that represents the placeholder. See “Adjusting the contents” on page 449 for details on the functionality available with Open Contents.

- *Variable:* If you select this option, when you “open contents” you will be able to change the size of the white rectangle without affecting the original placeholder. The portion of the drawing contained within the white rectangle is scaled to fit in the placeholder. See “Adjusting the contents” on page 449 for details on the functionality available with Open Contents.
- *On paper:* The scale ratio that you want to use on a paper copy of the layout.
- *Real World*: The “real” measurement for the scale ratio that you specify in the *On paper* box. For example, you may want to use a scaling of one inch in a paper copy of your layout. This one-inch measurement translates to a “realistic” measurement (for example, 3 feet).
- *Point Size*: The size at which points are printed.
- *Hide Zero Intensity Fixture*: Select this check box to hide any fixtures with an Intensity set to zero so they do not print.

View type tab

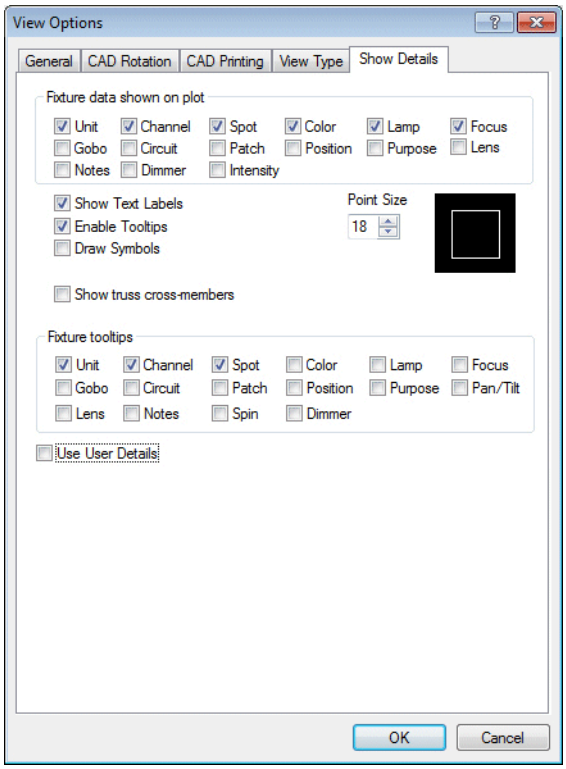
Options on the *View type* tab affect the view type of the CAD item.



- *View Type*: The view types available are those discussed in “Plot types” on page 104.
- *WorkPlane*: The workplanes available are those discussed in “Plot types” on page 104.
- *Scrollbars*: Select this check box to display the scrollbars on the bottom and left side edges of the layout. Click to clear this check box to turn the scrollbars off.

Show Details tab

Options on the *Show Details* tab affect the amount of fixture information that is shown in the CAD item.



- *Fixture data shown on plot*: Select the attributes that you want displayed around the fixture symbols in the CAD item. To modify how the attributes are laid out, refer to Fixture Notation and Information Layout in “Fixture notation” on page 308.
- *Show Text Labels*: Select this check box to display text labels.
- *Tooltips:* Displays tooltips on the CAD item. Tooltips are the pop-up messages that display specific information about a fixture when you hover over it. Choose the information that you want to display in a tooltip by selecting the appropriate check boxes in the *Fixture tooltips* group box. Tooltips are often helpful when trying to locate a specific fixture in a plot.
- *Draw Symbols:* Select this check box to draw fixtures as symbols instead of 3D objects. For more information on symbols, refer to “Symbols” on page 308.
- *Show truss cross members*: Deselect this check box to hide truss cross members in the CAD item to reduce the clutter or increase the speed of the display. Select this check box to see the cross members.
- *Point size:* Set the default size for all points. You can view a sample of the chosen size in the box to the right.

Adjusting the contents

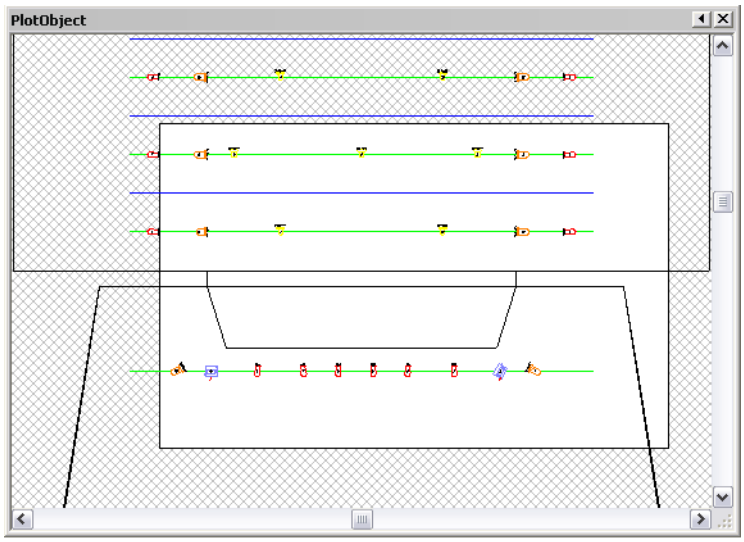
If you chose fixed or variable scaling on the *CAD Printing* tab or if you inserted a New Plots item, you can further set up how the item will appear within the placeholder by modifying the open contents.

To adjust the contents

1 With the item selected, from the **Edit** menu, choose **Open Contents**.

Tip: You can also click the *Open Contents* tool on the *Layout Pres Edit* toolbar.

Result: A new window opens, displaying your drawing and the item placeholder.



2 Adjust the positioning of the placeholder as desired.

Tip: To adjust the view of the contents, you can use your mouse wheel to zoom in or out or pan around by holding down the middle mouse button.

3 Close the window.

New Plot items

New Plot items are inserted into your layout using the options specified when you created the item. You might want to modify these options if you wish to display or print the New Plot item in black and white, for example.

To modify content properties of New Plot items

1 Select the New Plot item on the layout.

2 From the **Edit** menu, choose **Content Properties**.

Tip: You can also click the *Content Properties* tool on the *Layout Pres Edit* toolbar.

Result: The *View Options* dialog box is displayed.

For an explanation of the options that are available, refer to “Modifying the properties of a plot” on page 419.

Note: Any options that you modify here are also updated in the New Plots view that you inserted.

Image items

Special considerations should be given to printing images on layouts. To maintain the integrity of the image, WYSIWYG can maintain the aspect ratio of the image so as not to distort it.

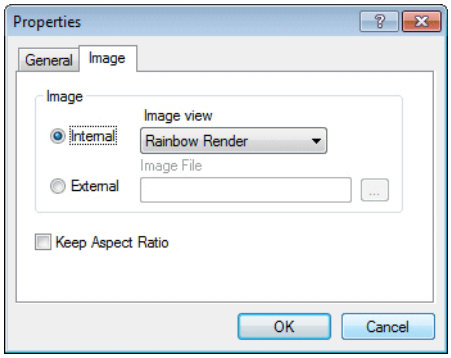
To maintain aspect ratio

1 Select the image item on the layout.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Layout Pres Edit* toolbar.

3 Click the *Image* tab.



4 Select the *Keep Aspect Ratio* check box.

Result: When this box is checked, the height versus width relationship of the original image is maintained regardless of how you resize the placeholder.

Worksheet, Report, and Spreadsheet items

Worksheet, Report, and Spreadsheet items are inserted into your layout at a set scale (100%) regardless of the paper size. For readability purposes, you can readjust this value.

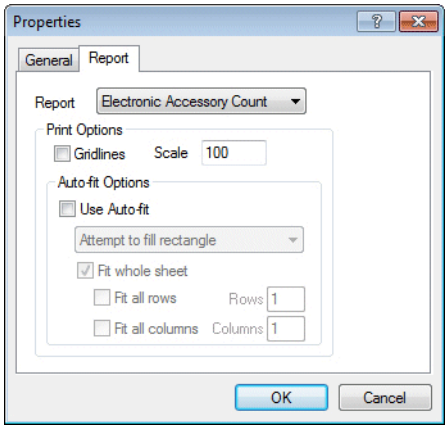
To adjust the properties of Worksheet, Report, and Spreadsheet items

1 Select the worksheet, report, or spreadsheet item on the layout.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Layout Pres Edit* toolbar.

3 Click the *Worksheet*, *Report*, or *Spreadsheet* tab as appropriate.



4 From the *Worksheet* drop-down box, select the item that you want to modify. The options vary based on the type of item that you have selected.

5 Click to select the *Gridlines* option to print gridlines.

6 In the *Scale* box, type a percentage value to increase or decrease the size of the contents.

7 Click *Use AutoFit* to automatically fit the item in the bounding box, and then choose an auto-fit option:

- *Attempt to fill*: WYSIWYG will attempt to fill the bounding box, ignoring the scale specified in the *Scale* box.
- *Limit*: WYSIWYG will still attempt to fill the bounding box; the scale specified in the Scale field will be considered.

8 Click to select *Fit Whole Sheet* to specify that AutoFit should apply to the entire contents of the worksheet/spreadsheet/report

9 Click to toggle *Fit all rows* and *Fit all columns* to specify that AutoFit should apply to all rows or all columns of the worksheet/spreadsheet/report. Alternately, you can specify to which rows and columns it should apply.

10 Click *OK*.

Legends

You may want to add a legend to your layout to provide a definition of the symbols used in the layout. To add a legend to a layout, you must first use the Legend Wizard to create the legend. The legend that you create can consist of the symbol name, wattage, count, and any other special notes. You can create multiple legends per layout.

To create legend items

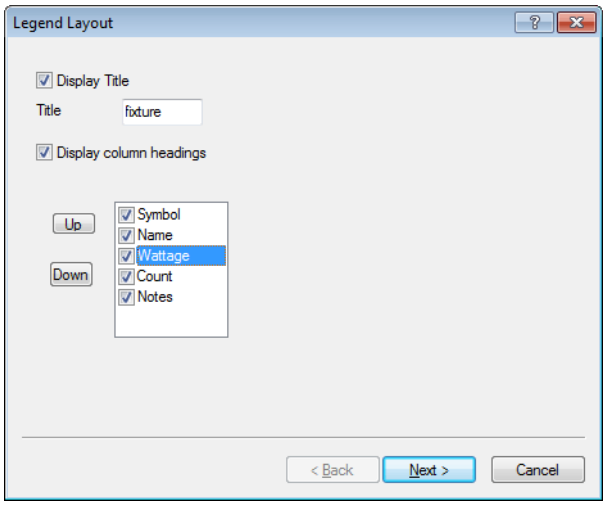
1 Right-click in the *Wizards* shortcut bar, and then select *New Legend*.

Result: The *Enter new Legend name* dialog box is displayed.

2 Type a name for the legend, and then click *OK*.

Result: The legend shortcut is stored in the shortcut bar and the Legend Wizard opens.

Step 1 - Legend Layout

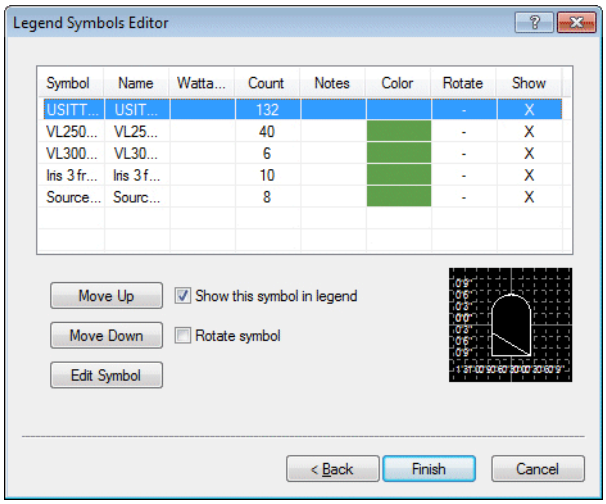


Options in Step 1 affect the layout of your legend.

a. *Title*: Select the *Display Title* check box if you want to display the name of the legend. Type a title in the *Title* box if you want a title other than the default name to be displayed.

b. *Columns*: Select the *Display column headings* check box if you want columns to be displayed in the legend. Choose the columns that you want displayed by selecting the appropriate columns from the list. The *Symbol* and *Name* columns are displayed by default. Order the display of columns by clicking the *Up* and *Down* buttons.

Step 2 - Legend Symbols Editor



Each symbol type that exists in your CAD drawing is displayed for possible inclusion in the legend. The counts of each symbol type are updated automatically as fixtures with the same symbol type are added or deleted from the CAD drawing.

Options in Step 2 affect the symbol information that is displayed in the legend.

a. *Show this symbol in legend*: By default all symbols are displayed in the legend. Click to deselect this check box if you do not want the highlighted symbol in the table to be displayed in the legend.

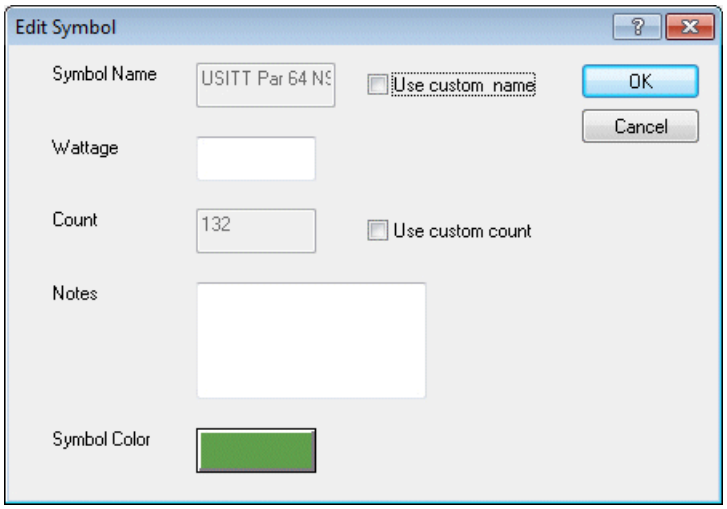
Order the display of symbols by clicking the *Move Up* and *Move Down* buttons.

b. *Rotate symbol*: Select this check box if you want the symbol to be rotated in the legend. The symbol will be rotated 90 degrees.

c. *Edit Symbol*: Click if you want to make modifications to the symbol information that is displayed (name, wattage, count, notes, color, etc.). Follow the steps in “Editing Symbols” to perform the modifications.

Editing symbols

When you click *Edit Symbol*, the following dialog box appears:



a. *Use custom name*: By default, the name displayed in the *Symbol Name* box is used, unless otherwise modified. To enter an alternative name, select the *Use custom name* check box, and then type the name.

b. *Wattage*: The wattage in volts for the lamp.

c. *Use custom count*: By default, the value displayed in the *Count* box is used, unless otherwise modified. This value includes only hung fixtures in your show file. To enter an alternate count, select the *Use custom count* check box, and then type the value.

d. *Notes*: Type any pertinent notes for the symbol.

e. *Symbol Color*: Click the *Color* box to open a color picker. Select the new color that you want to apply to the symbol, and then click *OK*.

3 When you are finished editing the symbol information, click *OK*.

Result: The *Legend Symbols Editor* dialog box is redisplayed.

4 Finish entering all relevant information, and then click *Finish*.

Result: The Wizard shortcut is stored in the shortcut bar with the name you specified in the *Title* box.

Editing a legend

To make changes to the information that is displayed in the legend, open the Legend Wizard by double-clicking on the legend shortcut, and then make any necessary modifications.

Once a legend has been defined, it does not change until you make modifications to its settings. If any new fixtures with different symbol types are added to the file since you last made changes, these additions will be available in Step 2 of the Wizard, but will remain deselected until you specify whether they should be displayed in the legend. The counts of symbol types that were already in the legend are updated (unless you opted to use a custom count).

To rename, delete, or clone the shortcut, follow the steps defined in “Working with shortcuts” on page 56.

Inserting a legend onto the layout

Once the legend has been created using the Legend Wizard, you can place it onto the layout.

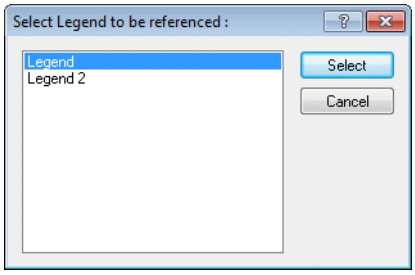
To insert a legend onto a layout

1 From the **Insert** menu, choose **Legend**.

Tip: You can also click the *Legend* tool in the *Layout Pres Insert* toolbar. 

2 Click and drag a window (placeholder) on the page where you want the item to appear.

Result: The *Select Legend to be referenced* dialog box is displayed.



3 Select the legend that you want to add to the page, and then click *OK*.

Result: The legend is added to the page at the position where you inserted the placeholder.

Modifying the properties of a legend

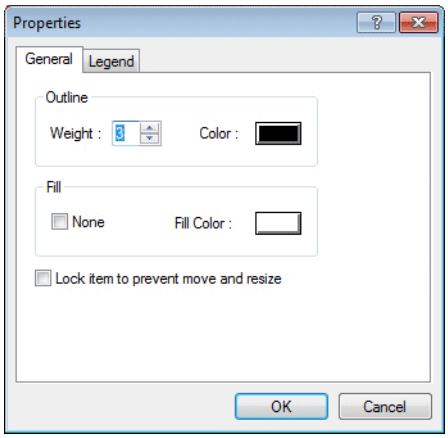
To modify the properties of a legend

1 On the layout, select the legend that you want to modify.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Layout Pres Edit* toolbar. 

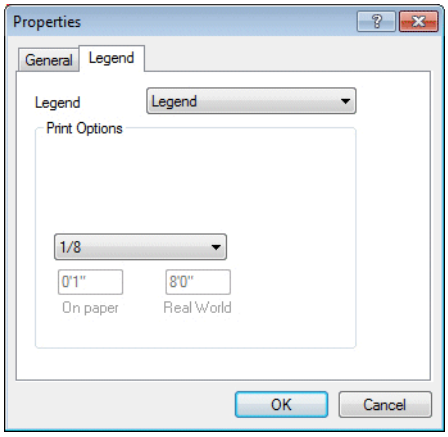
Result: The *Properties* dialog box is displayed.



The *General* tab is similar for all items. For more information on this tab, see “To modify item properties” on page 445.

Legend tab

Options on the *Legend* tab affect the scaling and content of the placeholder.



- *Legend*: Select the legend that you want to be displayed on the layout. A list of all the legends you have created is displayed in the drop-down list.
- *Print Options*: The scale ratio in inches and feet at which the legend should be printed. If the scale that you want to use for printing is not displayed, select *Custom*.
- *On paper:* The scale ratio that you want to use on a paper copy of the layout.
- *Real World*: The “real” measurement for the scale ratio that you specify in the *On paper* box. For example, you might want to use a scaling of one inch in a paper copy of your layout. However, this one-inch measurement translates to a more “realistic” measurement (for example, 3 feet).

Keys

You may want to add a key to your layout to provide a definition for the fixture notation used in the layout. To add a key to a layout, you must first use the Key Wizard to create the key. The key you create consists of the symbol of one given fixture and identifiers for the various attributes notated around the symbol. You can create multiple keys per layout.

To create key items

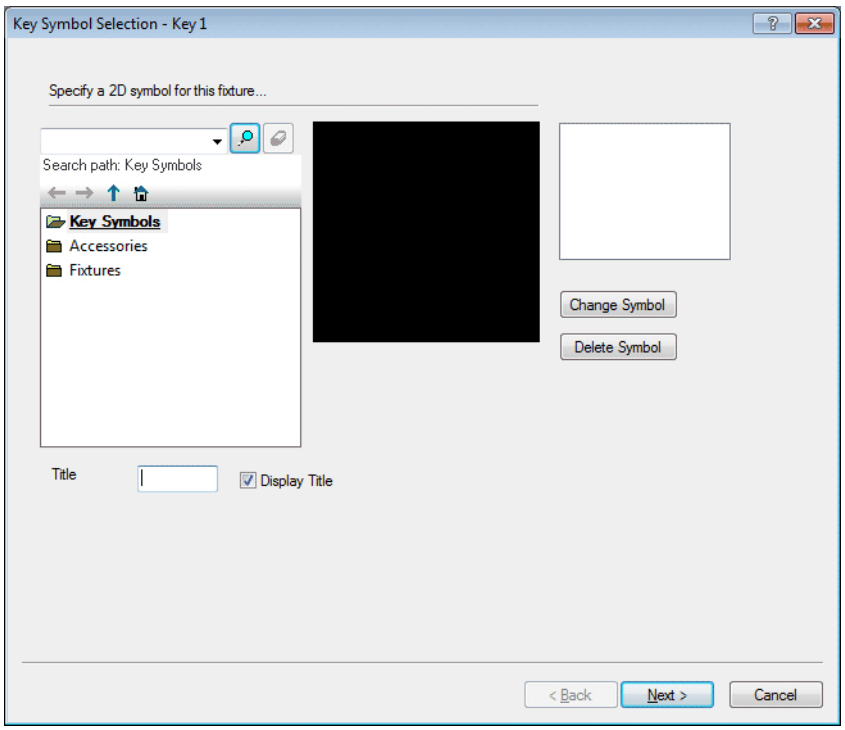
1 Right-click in the *Wizards* shortcut bar, and then select *New Key*.

Result: The *Enter new Key name* dialog box is displayed.

2 Type a name for the key, and then click *OK*.

Result: The key shortcut is stored in the shortcut bar and the Key Wizard opens.

Step 1 - Key Symbol Selection

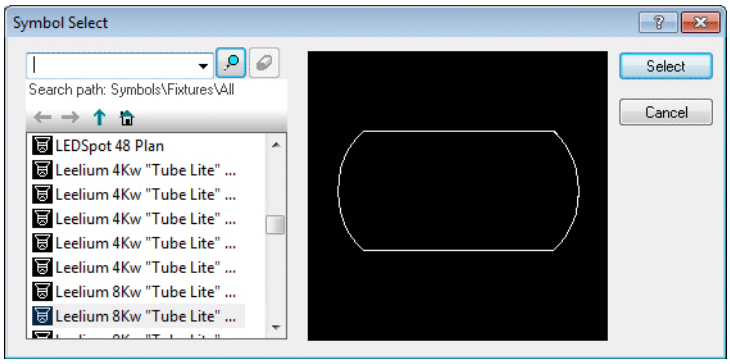


Options in Step 1 allow you to choose the symbol that will represent the fixture or accessory in the key.

- *List of symbols*: Select the symbols you want to use. You can choose up to one fixture symbol and one accessories symbol. The symbols available in this list are the default WYSIWYG symbols for each fixture or accessory type. When you selecting a symbol, it appears in the preview window to the right.
- *Title*: Select the *Display Title* check box if you want the name of the symbol to be displayed in the key. Type a title in the *Title* box if you want a title other than the default name to be displayed.
- *Change Symbol*: Enables you to choose a different symbol from the more extensive symbols library which includes the USITT standards, among others. Follow the steps in “Changing Symbols” to perform the modifications.
- *Delete Symbol*: Click this button to remove the selected symbol from the preview window.

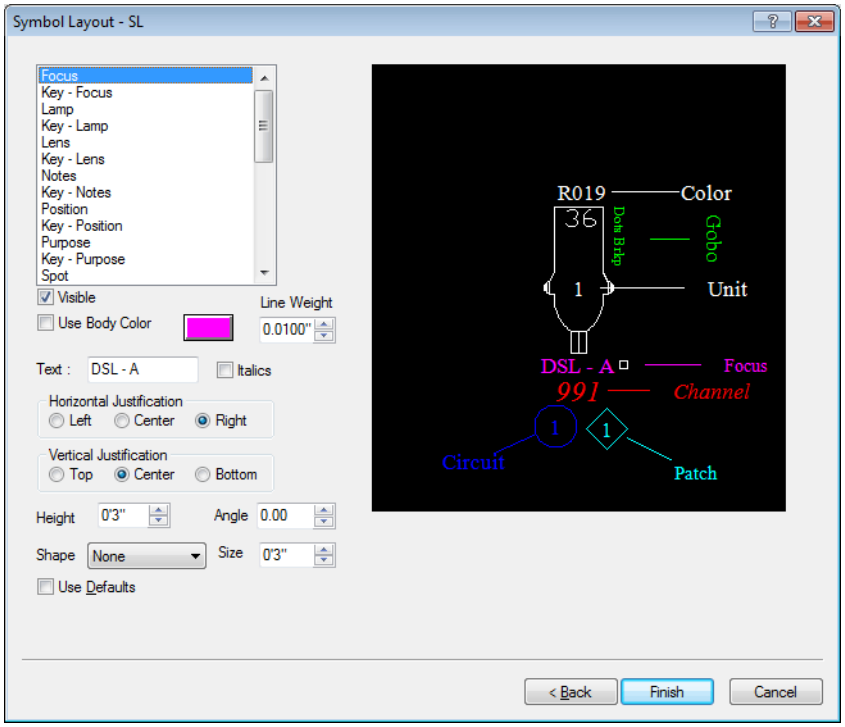
Changing symbols

From the list of items in the window to the right of the preview window, select the symbol. To change the symbol, click the *Change Symbol* button. The following dialog box appears:



From the list on the left side, choose the new symbol. When you are finished making your choice, click *Select*.

Step 2 - Symbol Layout



For each attribute there are two entries in the list box. One is for the attribute itself; the other is to label the attribute. The label is distinguishable by the qualifier "Key" (for example, the attribute *Focus* and the label *Key - Focus*). An extension line is drawn between the label and the attribute. For each attribute and its label you must decide whether you want to display it in the key. If yes, you can further specify how it will be displayed using the various formatting options (including text size and color, shape size and color, and justification).

You can also edit all the text, choose to give examples of attribute values, and provide custom definitions for the labels.

Options in Step 2 allow you to choose the attributes and notation to be displayed around the chosen symbols.

Note: To make the controls active, you may have to click to clear the *Use Defaults* check box.

- *Visible:* Select this check box if you want the attribute or label to be displayed around the symbol in your key.
- *Use Body Color:* Select this check box if you want the attribute or label to use the same font color as the symbol. To set a specific color for the attribute or label, leave the box unchecked and click the color box to choose a custom color.
- *Line weight*: Adjust this value to increase or decrease the density of the label or attribute font.
- *Text*: Type the text that you want displayed for the attribute or label.
- *Italics:* Select this check box to italicize the attribute or label font. Note there are no options to change the font.
- *Horizontal* and *vertical justification*: Choose the justification for the text. This is in relation to the insertion point or shape.
- *Height*: Increase or decrease to modify the font size.
- *Angle*: Specifies the attribute or label alignment in relation to the fixture symbol.
- *Shape*: Associate a shape with the label or attribute, if desired. You can choose from: none, circle, square, octagon, or diamond.
- *Size*: Increase or decrease the size of the shape as desired.

3 Set the position of the attribute or label by clicking and dragging on the label or attribute in the preview window. You can also select an attribute by clicking on it in the preview window.

4 The key as it will be drawn on the layout is displayed in the preview window. Once all the attributes are set as desired, click *Finish*.

Result: The Wizard shortcut is stored in the shortcut bar with the name you specified in the *Title* box.

Editing a key

To make changes to the information that is displayed in the key, open the Key Wizard by double-clicking on the key shortcut, and then make any necessary modifications. Once a key has been defined, it does not change until you make modifications to its settings.

To rename, delete, or clone the shortcut, follow the steps defined in “Working with shortcuts” on page 56.

Inserting a key into a layout

Once the key has been created using the Key Wizard, you can place it on the page.

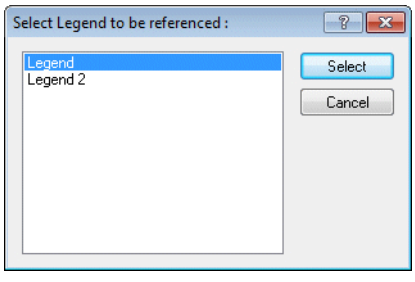
To insert a key into a layout

1 From the **Insert** menu, choose **Key item**.

Tip: You can also click the *Key* tool in the *Pres Layout Insert* toolbar. 

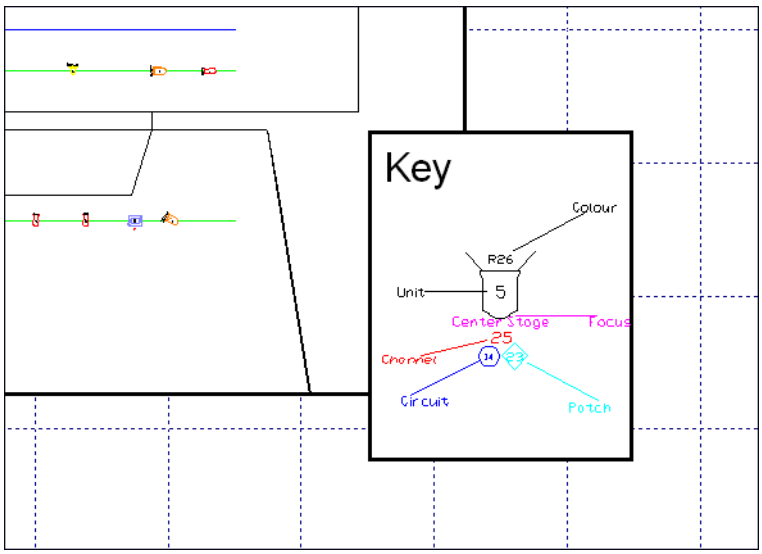
2 Click and drag a window (placeholder) on the “page” where you want the item to appear.

Result: The *Select Key to be referenced* dialog box is displayed.



3 Select the key that you want to add to the layout, and then click *OK*.

Result: Your key appears within the placeholder as drawn. By default key items are set to a scale of 1/16" = 1'.



Modifying the properties of a key

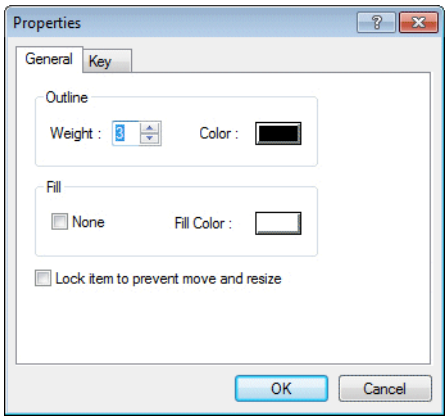
To modify the properties of a key

1 On the layout, select the key that you want to modify.

2 From the **Edit** menu, choose **Item Properties**.

Tip: You can also click the *Item Properties* tool on the *Pres Layout Edit* toolbar. 

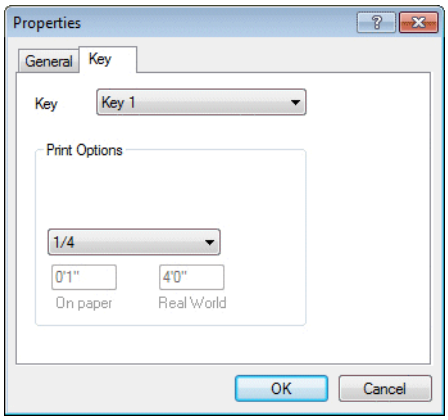
Result: The *Properties* dialog box is displayed.



The *General* tab is similar for all items. For more information on this tab, see "To modify item properties" on page 445.

Key tab

Options on the *Key* tab affect the scaling and content of the placeholder.



- *Key*: Select the key that you want to be displayed on the layout. A list of all the keys that you have created is displayed in the drop-down list.
- *Scale*: The scale ratio in inches and feet at which the key should be printed. If the scale that you want to use for printing is not displayed, select *Custom*.
- *On paper:* The scale ratio that you want to use on a paper copy of the layout.
- *Real World*: The "real" measurement for the scale ratio that you specify in the *On paper* box. For example, you might want to use a scaling of one inch in a paper copy of your layout. However, this one-inch measurement translates to a "realistic" measurement (for example, 3 feet).

Layout Tools

Locking items in Layouts

You can "lock" items in your Layouts to avoid accidentally moving or deleting them once you are satisfied with their placement. Once an item is locked, you can still select it (and access its Properties), but you can't move or delete it.

To lock an item in a Layout

1 In your Layout, click the item that you want to lock.

2 On the *Layout Pres Edit* toolbar, click to select the Lock icon. 

Note: Alternately, you can click CTRL + L to lock the selected item.

Result: The grips of the item turn red instead of black and you cannot move or delete the object.

To unlock an item in a Layout

When you click on an item in a Layout and see that its grips are red, or you notice that it cannot be moved or deleted, this means that the item is "locked." Follow the procedure below to unlock it.

1 In your Layout, click the item that you want to unlock.

2 On the *Layout Pres Edit* toolbar, click to select the Lock icon. 

Note: Alternately, you can click CTRL + L to unlock the selected item.

Result: The grips of the item turn black instead of red and you can now move or delete the object.

Placeholders

You can move and resize placeholders at any time.

To resize a placeholder

1 Click to select the presentation item.

2 Click and drag the marker points of the placeholder to resize it.

Note: Resizing a placeholder affects the content that is displayed. You might have to edit the scale of the contents or revisit open contents.

To move a placeholder

1 Click to select the presentation item.

2 When a small grey rectangle appears on your cursor, click and hold the left mouse button and drag the placeholder to the desired location.

3 Release the mouse button.

Tip: Use the gridlines of the presentation view to line up placeholders.

To copy a placeholder and its contents

1 Click to select the presentation item.

2 When a small grey rectangle appears on your cursor, press and hold the CTRL key.

3 Click and hold the left mouse button and drag the copied item to the desired location.

4 Release the mouse button.

Ordering

To facilitate overlaying and underlaying placeholders, use the *Bring to Front* and *Send to Back* tools. Think of placeholders as being piled on top of each other. *Bring to Front* moves the selected placeholder and its contents to the top of the pile. *Send to Back* moves the selected placeholder and its contents behind all of the others.

To overlay a placeholder

1 Click to select the presentation item.

2 From the **Edit** menu, choose **Bring to Front**.

Tip: You can also use the *Bring to Front* tool on the *Pres Layout Edit* toolbar 

To underlay a placeholder

1 Click to select the presentation item.

2 From the **Edit** menu, choose **Send to Back**.

Tip: You can also use the *Send to Back* tool on the *Pres Layout Edit* toolbar. 

Printing layouts

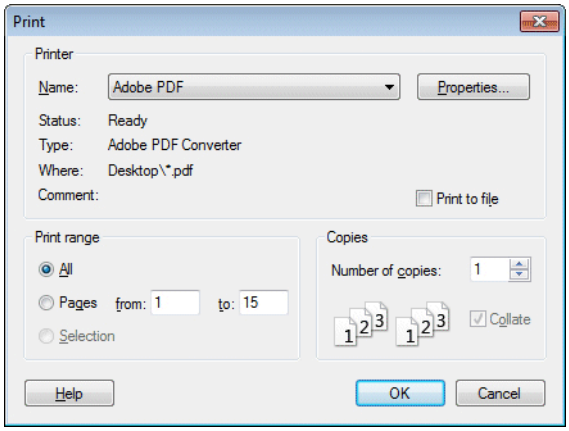
If you have a large layout you can “tile print” the layout across multiple smaller sheets of paper. To set up how you want the layout arranged on the sheets of paper, see “Working with layouts” on page 437.

You can print a complete layout or specific pages of a layout based on the settings that you choose when printing.

To print a layout

1 From the **File** menu, choose **Print**.

Result: The *Print* dialog box is displayed.



2 From the *Name* drop-down list, choose the printer (or plotter) that you want to use. WYSIWYG records the name that you choose for future reference and thereafter displays it as the default printer for layouts.

3 To print all pages of a layout, select *All*.

4 To print select pages of a layout, click *Pages*, and then enter the range in the *from* and *to* boxes.

5 If you want to print multiple copies of the pages, select the value from the *Number of copies* list box.

6 Click *OK*.

Batch printing

You can print multiple shortcuts at once on the Reports, Layouts, or Worksheets tabs in PRES mode, choosing to save them as PDF files in the folder of your choice or send them as PDF files to the Luminair app.

1 On the Reports, Layouts, or Worksheets tab, click **File** > **Batch Printing**.

2 Select the shortcuts that you want to print.

- To print to PDF, select the *Print to PDF* checkbox, and then select the output folder.
- To send PDF layouts, reports, and worksheets to the Luminair app, click *Send to Luminair*.

3 Click *OK*.



Images

Introduction

Images such as renderings and logos are stored in the *Images* tab of Presentation mode. Renderings are automatically saved here when you select “internal” in the Output Properties/Step 2 of the Render Wizard. Any other image files (bitmap or jpeg format) may be imported.

To import a bitmap or a jpeg

1. Right-click on the *Images* shortcut tab, and then choose **New Image**.
2. Type the name of your image.
3. Click *OK*.
4. Click on the shortcut for your new image. The text “No Image Available” appears in the working area of the screen.
5. From the **File** menu, choose **Import**.
6. Use the browser to find the image file that you want to import.
7. Click *Open*.

To export a bitmap or a jpeg

1. Select the image that you want to export.
2. From the **File** menu, choose **Export**.
3. Use the browser to select a destination and a name for the image file that you want to export. Ensure that either bitmap (.bmp) or jpeg (.jpg) is selected in the *Save As Type* box.
4. Click *Save*.



Worksheets

Introduction

The Worksheet tab contains a spreadsheet much like many other spreadsheet programs. You can enter data and create mathematical functions here, as well as import and export data to other spreadsheet programs. This functionality is useful when you are creating spreadsheets for crew requirements, schedules, budgets, or any other show-specific data.

To import data

1 Select **File > Import**.

2 Select the file.

3 Click *Open*.

To export data

1 Select **File > Export > Worksheet**.

2 Enter the file name for your worksheet.

3 Click *Save*.

Note: For optimal compatibility with other WYG files, it is recommended that you export to the WYSIWYG Spreadsheet format (*.wss).

To export a Worksheet template

For large shows, you may want to save the formatting of your worksheet as follows:

1 Select **File > Export > Worksheet Template**.

2 Enter the file name for your worksheet.

3 Click *Save*.

To create a new Worksheet

1 Click the Worksheets shortcut tab.

2 Right-click in the worksheets shortcut area, and then choose **New Worksheet**.

3 Type the name of your new worksheet.

4 Click *OK*.

To insert Smart Cells

1 In your worksheet, select a cell.

2 From the menu, select **Worksheet > Insert Smart Cell**.

To insert pictures

1 In your worksheet, select a cell.

2 From the menu, select **Worksheet > Insert Picture**.

3 Select a photo in one of the supported formats (*.BMP, *.JPG, *.GIF, *.TIFF, *.PNG, *.TGA).

4 Click *OK*.

To sort data

1 Select a cell block that needs to be sorted.

2 From the menu, select **Worksheet > Sort > Ascending/Descending**.

Adding/removing rows and columns

To add a row

1 Select the cell above which you want to add a row.

2 From the menu, select **Worksheet > Add > Row**.

To remove a row

1 Select a cell in the row that you want to remove.

2 From the menu, select **Worksheet > Delete > Rows**.

To add a column

1 Select a cell to the left of which you want to add a column.

2 From the menu, select **Worksheet > Add > Column**.

To remove a column

1 Select a cell in the column that you want to remove.

2 From the menu, select **Worksheet > Delete > Column**.

Merging/Unmerging Cells

To merge cells

1 Select the block of cells that you would like to merge.

2 From the menu, select **Worksheet > Merge Cells**.

To unmerge cells

1 Select the block of cells that you would like to unmerge.

2 From the menu, select **Worksheet > Unmerge Cells**.

Zebra Striping

To enable Zebra Striping on your worksheet

1 From the menu, select **Worksheet > Zebra Striping**.

Note: Currently there is only one set of alternating colors that you can choose.

Hiding Headers

To hide the headers (the labelling of rows and columns)

From the menu, select **Worksheet > Hide Headers**.

Freezing Panes

Freezing panes will lock a series of cells to the top of the working area no matter how much you scroll.

To freeze panes

1 Select the rows you want to freeze.

2 From the menu, select **Worksheet > Freeze Panes**.

Note: You can unfreeze panes by clicking **Worksheet > Unfreeze Panes**.

Formatting Cells

Each cell has a configurable Type, Border, Alignment, Font, and Background. To access these options, select a cell (or block of cells), and then select **Worksheet > Format Cells**.

Pipe tapes

Introduction

WYSIWYG enable you to make “Pipe Tapes,” which are attached to a pipe at the beginning of a hang. The pipe tape indicates to the electricians where to hang the fixtures and displays pertinent information about the fixtures, such as circuit number, dimmer number, gel color, and so on, as specified by the user. This tool significantly increases the accuracy between the WYSIWYG plot and the real installation.

Note: Pipe tape printers are sold separately and are available through CAST Group.

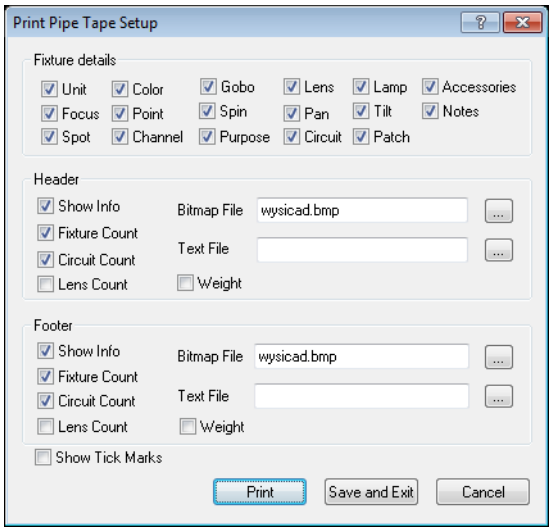
To print a pipe tape

1 Connect the pipe tape printer to your computer through the serial port.

2 In CAD mode, select the pipe for which you want to generate a pipe tape.

3 From the **Tools** menu, choose **Print Pipe Tape**.

Result: The *Print Pipe Tape Setup* dialog box is displayed.



4 In the *Fixture details* group box, select the data fields that you want printed on the pipe tape for each fixture.

5 To print show information such as venue, lighting designer, and show name, select the *Show Info* check box. Show information is entered in Show options, as described in “New Plots” on page 418.

6 To display a graphic in the header and/or footer, type a file name in the *Bitmap File* box or use the file browser to locate the file. To display correctly, the graphics selected must be monochrome bitmaps.

7 To print a total count of the fixtures on the selected pipe, select the *Fixture Count* check box. The total count is broken down by fixture type.

8 To print a total count of the circuits used by the fixtures, select the *Circuit Count* check box.

9 To display text in the footer and/or header of the pipe tape, type a text file name in the *Text File* box.

10 To print a total count of the number of lenses mounted in the fixtures, select the *Lens Count* check box. The total count is broken down by fixture type.

11 To print the weight of the fixtures on the pipe, select the *Weight* check box.

12 To display tick marks on the pipe tape, select the *Show Tick Marks* check box. Tick marks are placed every foot or half meter, based on the unit of measurement (imperial or metric). The tick marks are for reference purposes only.

13 When you have finished making your selections, click *Print* to send the print job to the pipe tape printer.

14 Click *Save and Exit* if you want to save the settings but do not want to print the pipe tape at this time.

Production Team Info Wizard

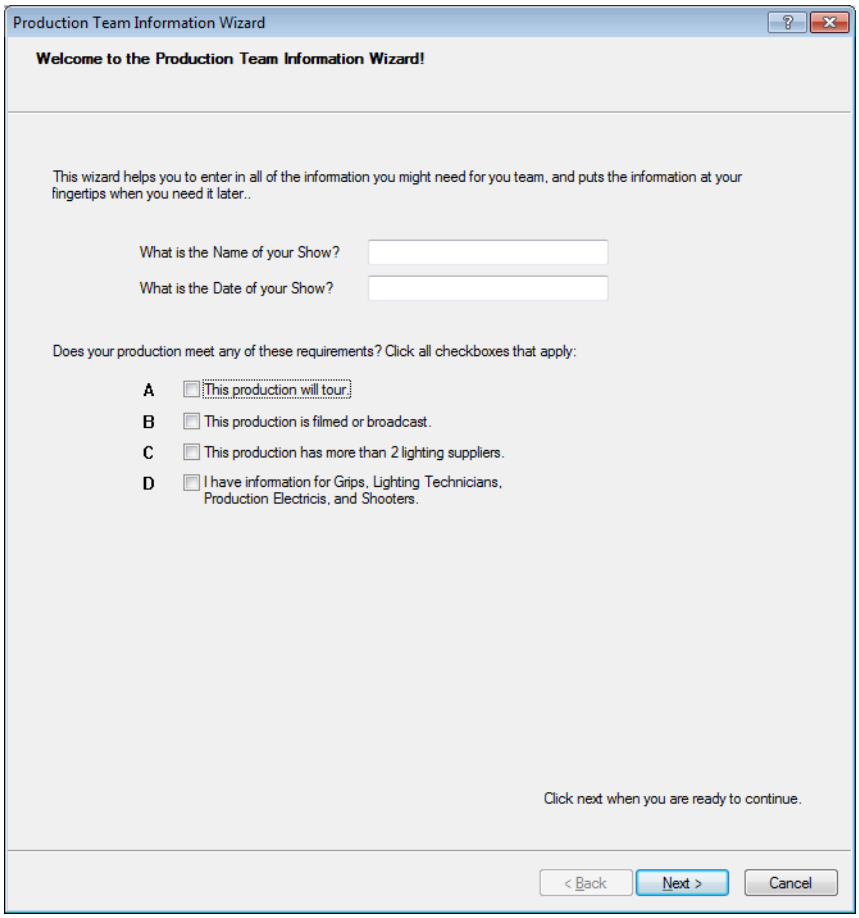
Introduction

The Production Team Info Wizard helps you gather and enter all of the information associated with your production team in one easy to access place.

To use the Production Team Info Wizard

1 In CAD mode, click **File** > **Production Team Info Wizard**.

Result: The Welcome window of the Wizard appears.



2 Type the name and date of your show in the boxes provided.

3 Use the checkboxes in the “Does your production meet any of these requirements?” section to specify whether you want to see extra information for entering team members:

- *This production will tour*: This option gives the user information on a touring production’s needs, such as Crew Chief, Lighting Director, and so on.
- *This production is filmed or broadcast*: This option gives the user extra fields for a typical film or television production crew.
- *This production has more than 2 lighting suppliers*: When your production is sourcing equipment from more than two lighting suppliers, this option gives you up to five contact field groups for your extra suppliers’ information.

- *I have information for Grips, Lighting Technicians, Production Electricians, and Shooters*: Check this box if you need to save contact information on the above types of production personnel.

4 Click *Next*.

5 Complete all the required information on each window of the Wizard, clicking Next to proceed to the next window.

6 When you reach the last screen, click Finish to save your information.

Result: You can access your information at any time through the Wizard itself (File > Production Team Info Wizard) or by choosing Options > Show Options > Production Team Info.

Chapter 9

Design mode



Design mode was created to provide lighting designers with an avenue for experimentation. Use it as a troubleshooting tool or as a creative tool to help you come up with cue concepts. In Design mode, you can create static lighting looks using the design tools, and then save and render those looks to output photorealistic pictures.

In this chapter

Layout tabs	473
Design tools	474
Fixture groups	484
Palettes	486
Looks	487
Cuelists	490
Camera Paths	493
Rendering	497
Background Rendering Manager	508



Layout tabs

Introduction

Beneath the work area in each mode is a series of layout tabs. These layouts provide various configurations of the views you are working with. To change layouts, click the tab that corresponds to the layout you wish to use. The Design mode contains the following layout tabs:

- Wireframe - The work area displays a full-screen wireframe view.
- Quad - The work area is divided into quadrants, three of which can be modified to show plan, front, side, or isometric views. The lower-right quadrant contains a shaded view.
- Shaded - The work area displays a full-screen shaded view.

Note: The wireframe views in Data mode are used for fixture selection only. To make any drawing modifications, you must return to CAD mode.

Design tools

Introduction

The design tools are available in the Design and Live modes. You must have your venue, scenery and fixtures drawn in order to use the design tools; however, you do not need to have all the data entered for the fixtures in Design mode. Design mode is intended so you do not need channel numbers or a patch. The design tools allow you to create virtual lighting looks without an external control console. You can rough-in static looks and render them for design presentations.

Though the Design mode looks a lot like the CAD mode, you cannot draw anything within the Design mode. If you need to make changes to your drawing, you need to switch over to the CAD mode.

You can see the output of fixtures in the wireframe and shaded views within the Design mode. However, you can only select fixtures within the wireframe views, unless you use Concept shortcuts as discussed in the next section.

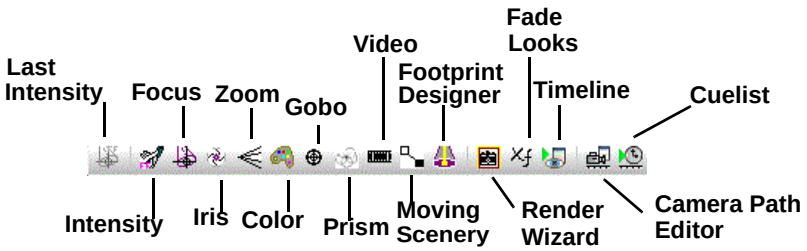
The design tools are individual toolbars that enable you to control certain parameters for selected fixtures. Currently there are ten design tools:

- Intensity
- Focus
- Iris
- Zoom
- Color
- Gobo
- Prism
- Video
- Moving Scenery
- Footprint

To turn the design tools on

1 From the **Design** menu, choose the tool name to activate or deactivate the desired design tool.

2 You can also toggle the tools on the *Design* toolbar.



Note: Design tools have the same properties as other toolbars and can be rearranged on the screen to improve the layout of the working area. You can also

resize design tools by stacking them into columns so that they take up less space on the screen.

To rearrange and resize design tools

1. Open the desired design tools using the steps above.
2. Click and drag one tool on top of another. When you release the mouse, the selected design tool lines up under the other.
3. Repeat step 2 for as many tools as will fit in a column.
4. If you overlay tools so that they are not completely visible, click on the triangle in the tools title bar. This expands or contracts the tool's window.
5. If you click and drag a tool's title bar off the column, it expands into its own window.

Using the Design tools

Begin by selecting the desired fixtures.

To select fixtures

Select a fixture (or fixtures). You can select multiple fixtures by holding the CTRL key while clicking on the fixture symbols. You can also use fixture groups as described in "Fixture groups" on page 484.

Tip: You can also right-click and drag a box around a section of your drawing to select fixtures. This displays a context-sensitive menu displaying all the fixture types contained within that box. You can then easily select "All Mac500," for example.

Result: The selected fixtures are highlighted in green.

Intensity tool

To use the intensity tool

The intensity tool provides many options for setting intensity levels.



Click the buttons for the extreme settings of intensity: *Full* (100%) and *Off* (0%).

You can quickly adjust the intensity by using the Plus or Minus buttons to alter intensity by 5%, 10% or 25%.

Use the dial to achieve any of the settings in between. To use the dial, click and hold the dial while you move the mouse up or down. The chosen intensity level is displayed in the intensity box as you move the mouse. Use the up/down arrows located above and below the dial to change the value by one.

Alternatively, you can enter a specific percentage by typing the number in the intensity box, and then pressing *Enter*. The beams will update instantly.

If multiple fixtures are selected, the intensity level entered is assigned to all fixtures. If you select multiple fixtures that have varying intensity levels, “N/A” appears in the intensity box.

The Fan feature enables you to select a group of fixtures, adjust the intensity as a group, and then spread the intensity out with a simple wheel. The fan allows for the distribution of intensity in the following configurations: Center, Left to Right, Right to Left, Chevron In, Chevron Out. Click the 'Fan' button to set the wheel in Fanning mode (color changes), and then move the wheel up or down to change fanning spread of the selected fixtures.

Zoom and iris tools

To use the zoom and iris tools

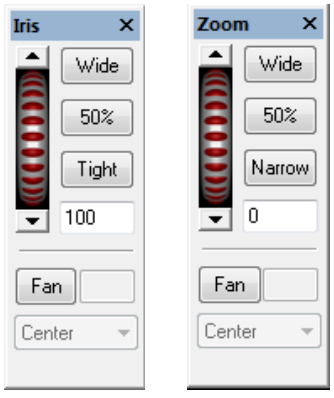
The zoom and iris design tools both work the same way.

Use the buttons for the extreme settings of tight or wide.

Use the dial to achieve any of the settings in between. To use the dial, click and hold the dial while you move the mouse up or down. The chosen value is displayed in the box as you move the mouse. Use the up/down arrows located above and below the dial to change the value by one.

Alternatively, you can enter a specific percentage by typing the number in the zoom or iris box, and then pressing *Enter*. The beam will update instantly.

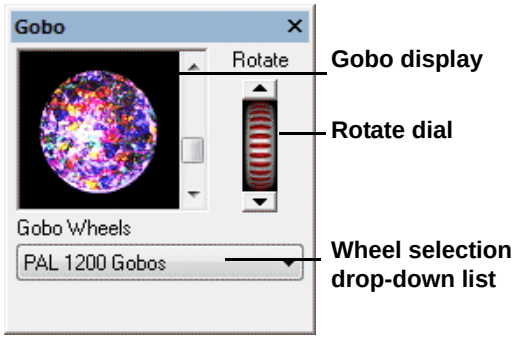
The Fan feature enables you to select a group of fixtures, adjust the zoom or iris as a group, and then spread the zoom/iris out with a simple wheel. The fan allows for the distribution of zoom/iris settings in the following configurations: Center, Left to Right, Right to Left, Chevron In, Chevron Out. Click the 'Fan' button to set the wheel in Fanning mode (color changes), and then move the wheel up or down to change fanning spread of the selected fixtures.



Gobo and prism tools

To use the gobo and prism tools

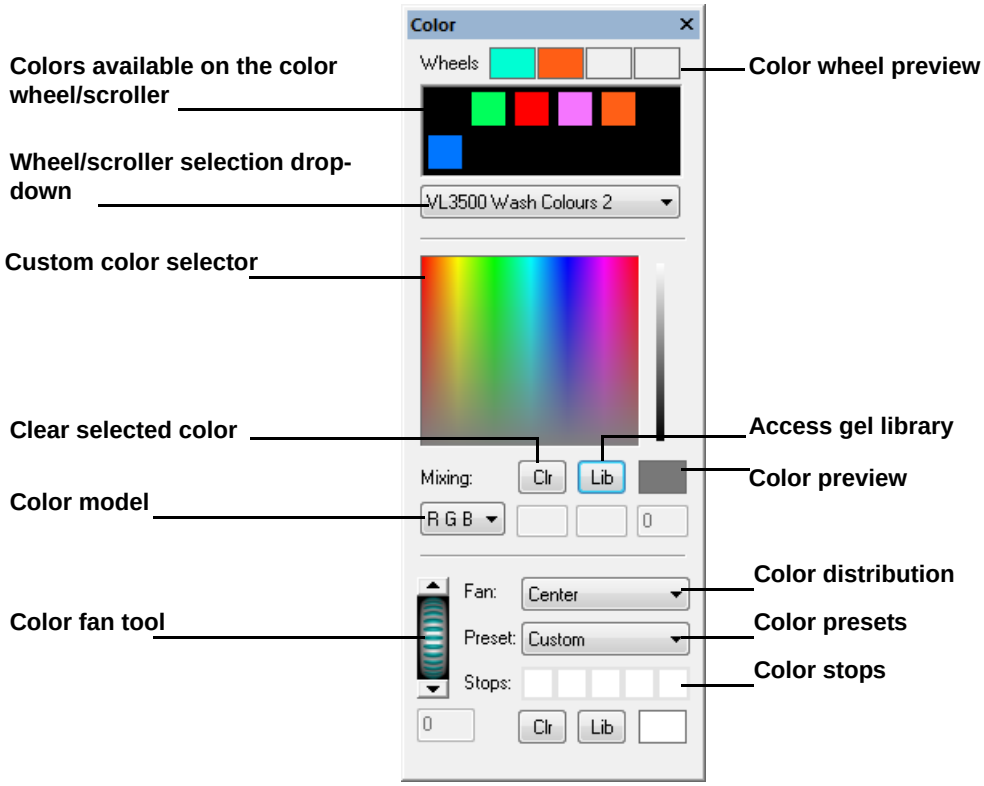
The gobo and prism tools work the same way. Specify the fixture type (if you have more than one selected), or the specific wheel to control (if that fixture type has multiple wheels) using the drop-down menu. Note that only fixtures with gobo or prism parameters appear on the list. Use the gobo/prism wheel scroll arrows to select the desired gobo/prism from the wheel. If no gobo/prism is desired, leave the wheel in the open slot.The dial will rotate this gobo if a rotating gobo is selected.



Color tool

To use the color tool

The color tool offers many options for color selection.



Click the custom color box or use the greyscale arrow to adjust your selection. The *Color model* values update automatically. The fixture's output depends on its capabilities. If the fixture has CMY or color mixing capabilities, the output is the exact color you selected. If the fixture is only equipped with a color wheel or a non-mixing scroller, the fixture will output the closest color to that selected, based on the available colors on the wheel.

The same principle works if you select a color from the library by clicking the *Library* button. The wheel/scroller selection drop-down list is as discussed for the gobo wheel. All of the available colors are displayed in the order in which they appear on the wheel or scroll.

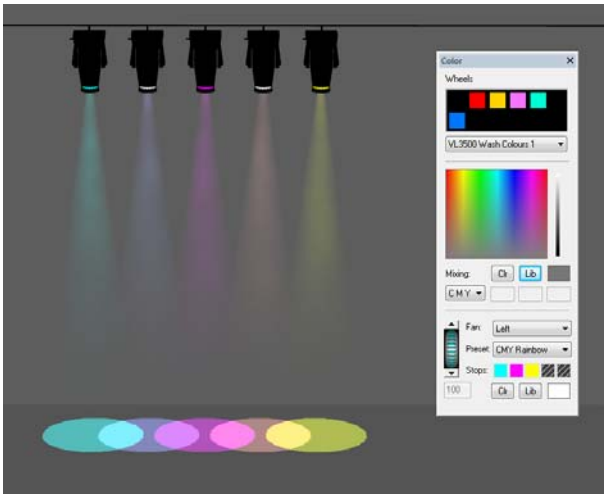
To facilitate designing with various fixtures using different color mixing techniques, the Color Designer tool also offers a drop-down with three different color space models (RGB, CMY, HSI). If you know the RGB values of the desired color, you can enter them, and the custom color box will update. Or if you have the RGB value and want to convert it to the CMY equivalent, change the Color Mixing selection.

You can also preview the color selected at all times using the Color preview box. Double-clicking on the Color Preview box has the same functionality as double-clicking on the Color Picker area, but without affecting your position on the area.

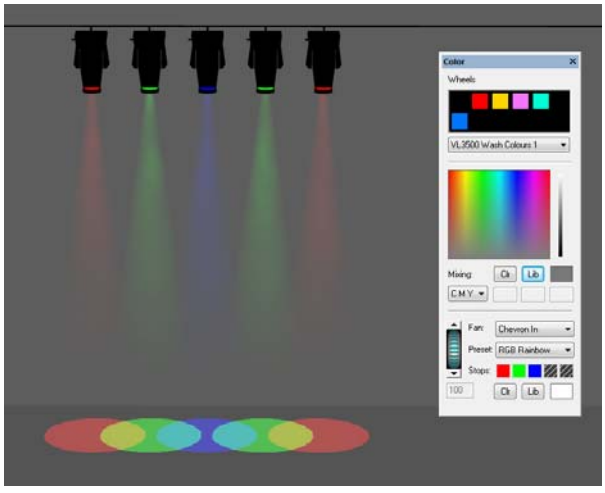
Note: These are valid value ranges for the color spaces:

RGB Color Space: R (0-255), G (0-255), B (0-255)
CMY Color Space: C (0-100), M (0-100), Y (0-100)
HSI Color Space: H (0-360), S (0-100), I (0-100)

The Color Fan feature enables you to select a group of fixtures, adjust the color as a group, and then spread the color out with a simple wheel. The fan allows for the distribution of color in the following arrangements: Center, Left to Right, Right to Left, Chevron In, Chevron Out.



Fan Left, CMY Rainbow configuration



Chevron In, RGB Rainbow configuration

Color Fan offers predefined Color Presets with defined color stops, or custom color stops can be created (limit 5 color stops) using colors chosen from the color selector or gel library.

Focus tool

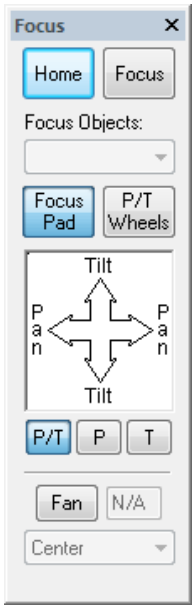
The focus tool works with fixtures that have variable pan and tilt parameters (i.e., automated fixtures). You must focus conventional fixtures in CAD mode.

Much like on a console, you can lock out Pan and/or Tilt from the Focus tool trackpad in order to prevent movement along a specific axis and help you focus your fixtures more effectively.

To use the Focus tool

The Focus tool gives you the option of using it in either Box or Wheel mode, as shown below:

Box mode



Wheel mode



From the *Pan and Tilt* drop-down box, choose whether you want to lock out either the Pan or Tilt direction. When you do so, you cannot move the fixture beams along the locked out axis. For example, if you select *Pan Only*, you cannot focus your fixtures along the Tilt axis. To enable focusing in all directions, leave the default *Pan and Tilt* selected.

Note: This setting applies only to the trackpad itself, and will not prevent you from clicking-and-dragging the beam in a Wireframe view or assigning a Focus Position.

In the focus trackpad, click, hold, and drag the mouse to control the position of the light beam. The *Home* button resets the fixture to pan = 50% and tilt = 50%.

To send the fixture's beam to a specific location, click the *Focus* button, and then click in your wireframe drawing. Use the focus position drop-down list to focus the beams of light toward one of the Focus Positions that you created in CAD mode.

You can also use the Focus tool to assign automated fixtures to focus objects that you have drawn in CAD mode. For more information on focus objects, see .

Note: The Focus tool does not take into account the current viewing position or fixture settings. In one orientation, dragging to the right on the trackpad results in the beam moving left, but rotating the fixture 180 degrees and dragging to the right will cause the beam to move to the right. This effect occurs with both moving mirror and moving head fixtures.

The Fan feature enables you to select a group of fixtures, adjust their focus as a group, and then spread the pan/tilt out with a simple wheel. The fan allows for the distribution of pan/tilt settings in the following configurations: Center, Left to Right, Right to Left, Chevron In, Chevron Out.

Click the 'Fan' button to set the wheel in Fanning mode (color changes), and then move the wheel up or down to change fanning spread of the selected fixtures.

Click and drag focusing

Instead of using the Focus tool, you can also focus fixtures by clicking on the beam in CAD Mode wireframe views and dragging the beam to the desired location.

To focus fixtures with click and drag

1 On the CAD Options toolbar, ensure that Toggle Beam Dragging is enabled. 

2 Turn the fixture on.

3 Click on the fixture's beam and drag it to the desired location.

To assign automated fixtures to focus objects

1 In CAD Mode, draw a Focus line by choosing **Draw > Focus <object>**.

Note: Any focus object can be drawn and have fixtures assigned.

2 In DESIGN mode, select the automated fixtures in the order in which you would like them to focus.

3 On the Focus Designer tool, from the Focus Position drop-down box, choose your focus object.

Video tool

Streaming video enables you to bring a live or pre-recorded video stream into WYSIWYG and play it back while you set looks in Design mode and while you precue in Live mode.

Notes:

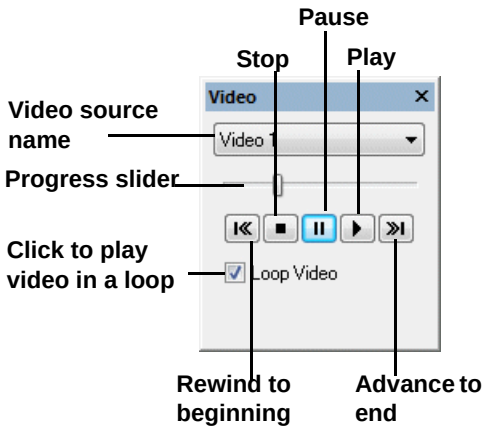
- The Video tool is also available in Live mode.
- You can only incorporate live video if you have WYSIWYG Perform; you cannot select a video from a live capture device or CITP/MSEX stream if you have WYSIWYG Design.

After you draw the screen and attach the video source or subsource to it (or to a 3D primitive, an LED Wall, or to a projector that you have inserted from the Fixtures library), you use the Video designer tool in Design mode to start the playback or stream. You can view the video in any of the shaded views, using the controls on the Video designer tool to pause, fast forward, rewind, or stop the video.

To use the Video tool

Once you have the video source or subsource attached to an object, you must use the Video designer tool to control it. For more information, see “To draw a screen” on page 163, “To configure a new video source for streaming video” on page 341, and “To create a single video subsource” on page 338.

Note: If the video control is patched and DMX connected in Live Mode, then you will not be able to control the video or video subsource using any of the Video designer tool commands in Design mode. Video sources and subsources can only be controlled by a designer tool when the DMX source is disconnected; therefore, you must first disconnect the applicable console device from within Live Mode before using any of the Video Designer Tool commands.



1. Click the Shaded tab to view the video screen(s) (or LED Walls or projectors) that you have drawn.
2. Click the Video designer tool icon to open the tool window.
3. From the drop-down list, select the video source or subsource.
4. Use the controls shown in the graphic above to play, pause, or stop the video. You can also use the slider to manually advance or rewind the video at your desired speed.

   Note: When you click the Pause button, the frame of the video that is playing at the time is held on the video screen; when you click the Stop button, the video stops playing and the screen goes blank.

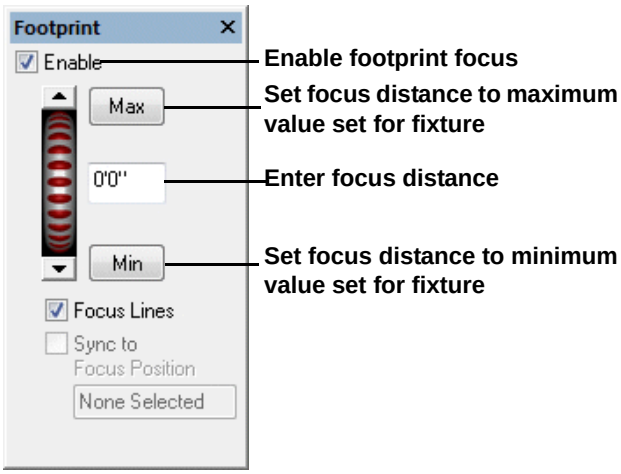
Footprint tool

Using the Footprint tool, you can change a footprint's focus distance for automated fixtures in the Shaded view of Design mode, in effect making the focus of the footprint "hard" or "soft".

Note: In this release of WYSIWYG, this feature is available in CAD and Design modes only: Footprint Focus is configurable for conventional fixtures in CAD mode and configurable for automated fixtures in Design mode (see page 293). Once configured, all fixtures can be visualized in Design mode. In an upcoming release, the Footprint Focus feature will be extended to function in Live Mode like all other Designer tools.

To use the Footprint tool

1 Select the automated fixture and open the Footprint Designer tool.



2 Check *Enable* to enable *Footprint Focus* for the selected fixture(s).

3 Adjust the focus distance using the slider or by entering a value in the *Focus Distance* box. Click the *Min* and *Max* buttons to set the value to the fixture's minimum or maximum focus distance, which are defined on the General tab of the fixture's Property page in the Footprint Settings section.

4 Check *Focus Lines* to display focus lines for the selected fixture(s) in shaded views.

5 Check *Sync to Focus Position* to adjust the footprint focus distance to the specified *Focus Position*.

The effect is visible in the Shaded view when *Footprint Focus* is enabled on the Lighting tab of the View Options window.

Moving Scenery tool

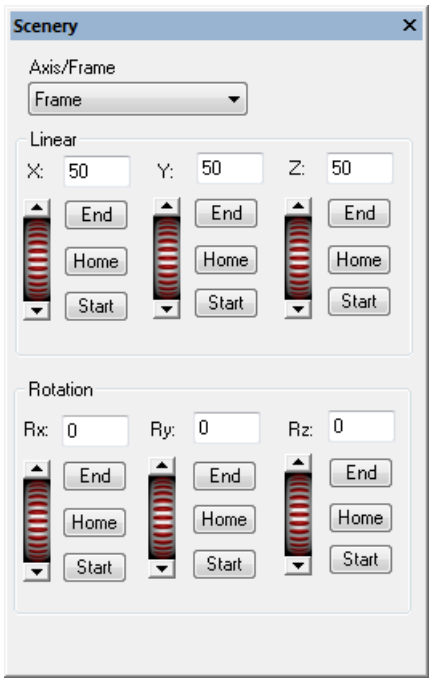
After drawing motion axes or frames and attaching objects to them, you can use the Moving Scenery Designer tool to make the object "move" along each axis.

You can simulate moving scenery by setting up looks containing motion axes or frames in Design mode. Once you create the look and specify its fade time, use the Moving Scenery Designer tool to customize the look by selecting the appropriate motion axis or frame and the object's starting position on it. Then create a second look, repeating the same procedure to set the object's starting position on either the same axis or frame, or a different one (if there is more than one axis attached together).

When you switch from one look to the next, you can see the object move from the starting point of the first look, to the starting point of the second look within the time period that you define as the "fade time".

To use the Moving Scenery tool

After drawing motion axes/frames and attaching objects to them, you can use the Moving Scenery tool to make the object "move" along each axis.



1 In Design mode, click the Moving Scenery Designer tool icon to open the tool.



Note: You can also open the tool by clicking **Design** > **Moving Scenery tool**.

2 From the drop-down list, select the appropriate motion axis or frame.

3 If you selected a linear axis, the X scroller is enabled. If you selected a rotational axis, the Rx scroller is enabled. If you selected a motion frame, all scrollers with activated motion ranges are enabled.

4 To move the object on the axis, you have four choices:

- Use the scroller to move the object back and forth along the axis.
- Click *Start* or *End* to move the object immediately to either the start or end of the motion axis.
- Click the *Home* button to move the object to its original position.
- For very precise positioning, in the box provided, type the object's position as a percentage of its total range of movement and press ENTER. For example, if you want to see the object at the exact half-way mark, type 50 in this box, and then press ENTER.

Fixture groups

Introduction

Fixture groups are custom groups of fixtures. You can create fixture groups in CAD, Data, Design, or Live mode in any view that allows fixture selection.

Creating fixture groups

Creating a fixture group allows you to select a group of fixtures at the same time.

Fixture group shortcuts also enhance working with design tools in shaded views. Since design tools work with the current fixture selection, opening multiple fixture group shortcuts allow you to change different groups of fixtures, rather than having to select them individually in the Wireframe view.

Note: Fixture groups are saved on the *Fixture group* shortcut bar.

To create a new fixture group

1. Select the fixtures that you want to group. If you are unfamiliar with selecting objects, refer to “Selecting” on page 185.
2. Click the Fixture groups shortcut bar.
3. Right-click on the Fixture groups shortcut bar, and then choose **New Fixture Group**. Alternately, you can click the new fixture icon  .
4. Type the name of the fixture group, and then click *OK*.

   Result: The fixture group is saved on the Fixture Group shortcut bar.

Updating fixture groups

You can modify a fixture group shortcut by adding or removing fixtures as required.

To update fixture groups

1. Select the set of fixtures that you want to include in the fixture group.
2. Right-click on the *Fixture Group* shortcut that you want to update, and then choose **Update**.

   Result: The *Fixture Group* shortcut is modified to include only the fixtures that you selected.

Working with fixture groups

When you select a *Fixture Group* shortcut, any currently selected objects are deselected, and the fixture group set is selected instead.

If you press the CTRL key while selecting fixture groups, the fixture group is added to the current selection set.

If you press the CTRL and SHIFT keys while selecting fixture groups, the fixture group is removed from the current selection set.

Tip: Fixture groups can be used to select fixtures even while in Shaded view.

Notes:

1. In a fixture group shortcut, multi-cell fixtures, such as cyc lights, can be selected either by cell or fixture. If the fixture group was created in Design mode using cells, the entire fixture is selected in the other modes. If the fixture group was created using fixtures, all cells are selected when in Design mode.
2. When you replace a multi-cell fixture that is selected by circuit with a different multi-cell fixture that has a different number of circuits, the replacement fixture is selected by fixture and not by circuit.
3. When you delete a fixture, the fixture is removed from all fixture groups that include that fixture.
4. When you replace a fixture using the **Replace Fixtures** menu option, any fixture groups referring to that fixture are updated to refer to the fixture replacement.
5. Any fixture added to a document will not initially be part of any fixture group.

Palettes

Introduction

You can create palettes of color to facilitate color selection. A palette is a reference to a specific color and can be used on all color-changing fixtures and scrollers.

Creating palettes

Creating a Palette shortcut is helpful if you want to save your favourite CMY color mixes for later use.

Note: You can create palettes in Design mode only.

To create a palette

1 Select the fixture from which you want to capture the color.

Note: You can create a palette from only one fixture.

2 Right-click on the *Palette* shortcut bar, and then choose **New Palette**.

3 Type the name of the palette, and then click *OK*.

Result: The CMY value is saved as a palette in the Palette shortcut bar. This value can be a CMY mix, a color from the color wheel or scroll, or a combination of color sources.

Updating palettes

You can modify a Palette shortcut by modifying or replacing colors as required.

To update a palette

1 Select the fixtures from which you would like to record the palette.

2 Right-click on the *Palette* shortcut that you want to modify, and then choose **Update**.

Result: The Palette shortcut is updated with the new CMY values.

To apply a palette

Applying a color palette is the equivalent of setting up color parameters for the selected fixtures, except it is much faster.

1 Select the fixtures for which you want to assign the specific color value.

2 Click on the desired palette shortcut.

Result: The fixtures are assigned the recorded copy value. If the fixture has CMY or color mixing capabilities, the output is the exact color that you selected. If the fixture is only equipped with a color wheel or a non-mixing scroller, the fixture outputs the closest color to that selected, based on the available colors on the wheel.



Looks

Saving lighting looks

Changes that you make with the design tools update the currently selected look. The name of the currently selected look is displayed at the top of the working area. Before making changes, always check that the currently selected look is the one that you want to modify.

To create a new lighting look

1. In the shortcuts bar, click *Looks*.
2. In the Looks shortcut area, right-click, and then select **New Look**.
3. In the *Name* box, type the name of the new look.
4. In the *Fade time* box, type the fade time in seconds for this look. This is the amount of time that it takes to "fade" to this look when you click on it from another look in the shortcut bar. For more information, see "Cross-fading between lighting looks" below.
5. Click *OK*.
6. Scroll to the bottom of the *Looks* shortcuts list.
7. Click on the shortcut for your new look.
8. Use the design tools to create your new lighting look.

   Tip: To build on an existing look without losing it, clone the look shortcut and work from the copy of the shortcut. For more information on cloning shortcuts, refer to "To clone a shortcut" on page 57.

Cross-fading between lighting looks

You can use the Looks shortcut bar to quickly set up and run timed transitions between lighting looks without the need of a lighting console. When you create a lighting look, you can specify its fade time in seconds. This is the amount of time that it takes to "fade" to this look when you click on it from another look in the shortcut bar. You can create multiple looks and specify different fade times for each of them.

Once you create the look and specify the fade time, you can use the design tools to customize the look. For example, you can add color, gobos, and intensity to lighting, and set the position of moving lights. When you switch from one look to the next, you can see the movement of the lights from one position to the next, along with any changes you have made between looks, such as color, intensity, and so on.

To cross-fade between looks

1. In Design mode, in the shortcuts bar, click *Looks*.
2. In the Looks shortcut area, right-click, and then select **New Look**.
3. In the *Name* box, type the name of the new look.
4. In the *Fade time* box, type the fade time in seconds for this look.
5. Click *OK*.
6. Scroll to the bottom of the *Looks* shortcuts list.

7 Click on the shortcut for your new look.

8 Use the design tools to create your new lighting look. You can set the color, intensity, zoom, iris, and add gobos. If you are using moving lights, you can also set the position of the lights.

9 To create the next look, in the Looks shortcut area, right-click, and then select **New Look**.

10 In the *Name* box, type the name of the new look.

11 In the *Fade time* box, type the fade time in seconds for this look.

12 Click *OK*.

13 Scroll to the bottom of the *Looks* shortcuts list.

14 Click on the shortcut for your new look.

15 Use the design tools to create your new lighting look. You can set the color, intensity, zoom, iris, and add gobos. If you are using moving lights, you can also set the position of the lights.

16 Ensure that the *Fade looks* button is enabled so the look fades instead of jumping directly to the next look. 

Tip: If the button is not enabled, then you can "jump" from one look to the next by clicking the look shortcuts in the shortcut bar. Even if the button is enabled, you can always jump to the next look by right-clicking the look shortcut, and selecting **Jump to**.

17 To watch the "fade" from the first look to the second look, click the shortcut for the second look. The image fades over the period of time that you specified for the second look.

Tips:

- For a more realistic view of the fading between looks, click the *Shaded* tab.
- To jump directly to a specific look, right-click the look, and then select **Jump to**.

Viewing moving scenery

You can use the Looks shortcut bar to quickly set up and run timed transitions between looks to simulate moving scenery.

When you create a look, you can specify its fade time in seconds. This is the amount of time that it takes to "fade" to this look when you click on it from another look in the shortcut bar. You can create multiple looks and specify different fade times for each of them.

Just as you can specify the fade time to watch the transition between lighting looks, so too can you use the fade time to simulate moving scenery. In the latter case, however, your Look must contain at least one motion axis with at least one object attached to it.

Note: You can also combine Looks to simulate both moving lights and moving scenery in one "fade".

Once you create the look and specify its fade time, use the Moving Scenery Designer tool to customize the look by selecting the appropriate motion axis and the object's starting position on it. Then create a second look, repeating the same procedure to set the object's starting position on either the same axis, or a different one (if there is more than one axis attached together).

When you switch from one look to the next, you can see the object move from the starting point of the first look, to the starting point of the second look.

To view moving scenery

Note: Before you perform this procedure, you must have drawn at least one motion axis and attached at least one object to it. For details, see “Drawing motion axes” on page 170.

1 In Design mode, in the shortcuts bar, click *Looks*.

2 In the Looks shortcut area, right-click, and then select **New Look**.

3 In the *Name* box, type the name of the new look.

4 In the *Fade time* box, type the fade time in seconds for this look.

5 Click *OK*.

6 Scroll to the bottom of the *Looks* shortcuts list.

7 Click on the shortcut for your new look.

8 Click the Moving Scenery Designer Tool icon ().

9 From the drop-down list in the Scenery window, select the appropriate motion axis.

10 Click the slider to advance the object to the position where you want it to start on the selected axis.

Note: You can also use the *Start* box to type the object’s position as a percentage of its full range of movement. For example, to show the object at the exact half-way mark on the motion axis, type 50.

11 To create the next look, in the Looks shortcut area, right-click, and then select **New Look**.

12 In the *Name* box, type the name of the new look.

13 In the *Fade time* box, type the fade time in seconds for this look.

14 Click *OK*.

15 Scroll to the bottom of the *Looks* shortcuts list.

16 Click on the shortcut for your new look.

17 From the drop-down list in the Scenery window, select the appropriate motion axis.

18 Click the slider to advance the object to the position where you want it to start on the selected axis.

19 Ensure that the *Fade looks* button is enabled so the look fades instead of jumping directly to the next look. 

Tip: If the button is not enabled, then you can “jump” from one look to the next by clicking the look shortcuts in the shortcut bar. Even if the button is enabled, you can always jump to the next look by right-clicking the look shortcut, and selecting **Jump to**.

20 To watch the “fade” from the first look to the second look, click the shortcut for the second look. The object moves from the starting point of the first look to the starting point of the second look over the period of time that you specified as the fade time for the second look.

Tips:

- For a more realistic view of the fading between looks, click the *Shaded* tab.
- To jump directly to a specific look, right-click the look, and then select **Jump to**.
- For details on controlling the object’s movement with a console device, such as the Mini Console, see “To control a DMX patched motion axis with a console” on page 532.

Cuelists

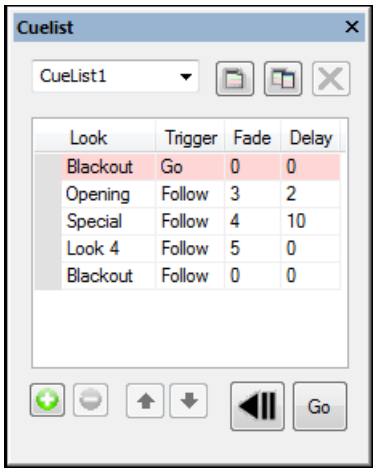
About Cuelists

Cuelists enable you to play back stored Looks in any order, without having to resort to the crossfade function described in the previous section.

To create a Cuelist

1 On the Design toolbar, click the Toggle Cuelist icon to open the Cuelist tool.

Result: The Cuelist tool appears.



2 In the top-right corner of the tool, click the *New* button.

3 Type a name for the Cuelist, and then click *OK*.

To rename a Cuelist

1 In the Cuelist tool, click the drop-down in the top left corner and select the Cuelist that you want to rename.

2 With the Cuelist highlighted, type the new name, and then press ENTER.

To copy a Cuelist

This procedure is useful when you set up one Cuelist with Looks in a certain order, but want to see how the same Looks would "play out" in a different order.

1 Click to select the Cuelist that you want to copy.

2 Click the Clone button in the top right corner.

To delete a Cuelist

1 Click to select the Cuelist that you want to delete.

2 Click the Delete button in the top right corner.

To add a cue to the Cuelist

1 To add a cue to a Cuelist, select the Cuelist from the drop-down in the top left corner.

2 Click the + button (lower right corner) to add a cue.

Note: To delete a cue, select it and then click the minus (-) button in the bottom right corner.

To adjust the position of a cue

1 To adjust the position of a cue in the list, click to select it, and then move it up and down using the arrow buttons in the lower left corner.

To assign a Look to a cue

1 Once you add a cue, you can assign a Look to it by clicking the drop-down in the Look column and selecting the appropriate Look.

Setting cue triggers

There are two types of triggers that you can set for each cue in your Cuelists:

- **Go**: The fade for the current cue will start immediately (the delay time is 0 seconds).
- **Follow**: The fade for the current cue will activate automatically after the set delay time has passed.

To switch between triggers, click to select the appropriate cue, and then click in the Trigger cell. From the drop-down, you can toggle between Go and Follow.

Setting the fade time

The fade is the time (in seconds) that it takes to transition between the previous cue and the current cue.

Notes:

- The fade time is in no way related to the crossfade time set for individual looks.
- The fade always starts after the delay time has passed.

To enter the fade time, click to select the appropriate cue, and then click in the Fade cell to type the new fade time.

Setting the delay time

The delay is the time (in seconds) that it takes for lights to *start* fading *into* the current cue. To enter the delay time, click to select the appropriate cue, and then click in the Delay cell to type the new delay time.

To activate a cue

1 To activate a cue, hit the *Go* button. Note that if the next cue has a 'follow' trigger, it will start as soon as its delay time has passed.

To stop playing a Cuelist

1 To stop playing a Cuelist, click the ||< button. Click the same button to step back through cues, disregarding fades and delays.

Tips: To "jump into" a cue (i.e., ignoring fade and delay times), double-click in the gray area to the left of the cue. You can use the arrow keys to navigate the Cuelist table.

Camera Paths

Introduction

Once you draw a Camera path in CAD Mode, you can use the Camera Path Editor and the Timeline tool in Design or Live Mode to define the time interval between two nodes along the path. If there are Camera Targets inserted in the file, you can also specify when the camera should follow a camera target. Finally, you can also define the orientation and field angle of the Camera at the current node position.

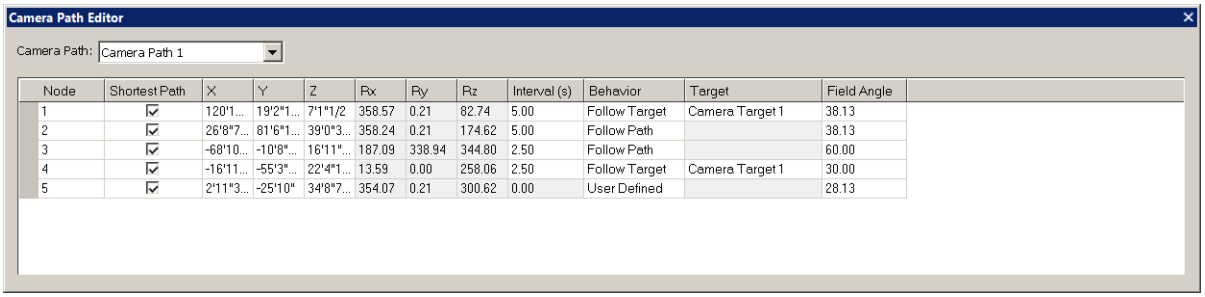
Once you have defined the time intervals and camera's orientation, you can open the Timeline tool and watch the Camera path play in the Shaded View. Camera Path playback is also simulated in Full Screen mode.

You can also patch a camera path to a DMX universe and then control the camera's movement through a console. For details, see "Patching camera paths" on page 550.

Using the Camera path editor

Once you create a Camera path in CAD mode, follow the steps in this section to define the time intervals between the nodes on the path, change camera behavior, assign targets to camera nodes, and reposition nodes in space. For details on drawing Camera paths, see "Drawing Camera paths" on page 167.

Camera path editor window



Camera path editor columns

- *Node*: This column gives you the Node number of the nodes on your Timeline.
- *Shortest Path*: When the Shortest Path checkbox is selected for a node, it forces the camera to rotate as little as possible when moving between two nodes on your timeline. When the Shortest Path checkbox is unchecked for a node, the camera will take the longer of the two directions as it rotates to aim at the next node's target. This applies to Roll, Pitch and Yaw.
- *X, Y, and Z Columns*: The X, Y, and Z columns in the Camera Path Editor are editable position information for each of the nodes on your timeline. You may enter in distance values into these fields to re-position your nodes as you see fit.
- *Rx, Ry, and Rz Columns*: These columns represent the rotation around each of the X, Y, and Z axes in WYSIWYG. These columns are non-editable, but allow you to get positional information from your nodes.
  - Rx, or Rotation around the X Axis = PITCH[tilting forward and backward]
  - Ry, or Rotation around the Y Axis = ROLL [tilting side to side]
  - Rz, or Rotation around the Z Axis = YAW [also called Heading or Direction]

- *Interval*: A node's Interval is how long it takes in seconds to get from one node to another. For example, if the timing on node 2 is ten seconds (10s), then the camera would move for ten seconds before reaching node 2.
- Behavior: Behavior is the camera's movement and direction behavior as it relates to the Camera Path. There are three types of Camera Behavior:
  - *Follow Path*: The camera will follow straight along the path, not deviating in any way possible from the path as it travels between nodes.
  - *Follow Target*: The camera will always point at an assigned Camera Target if this behavior is chosen for a node.
  - *User Defined*: This option allows the camera to behave exactly as specified by a user, meaning that the camera will point wherever you point it when the User Defined behavior is chosen.
- *Target*: Each node can be assigned a different target when the Follow Target behavior in the Behavior column is chosen. The Target column is a drop-down box for choosing different targets in your drawing.
- *Field Angle*: This option allows you to enter in different field angles for each node.

To use the Camera path editor

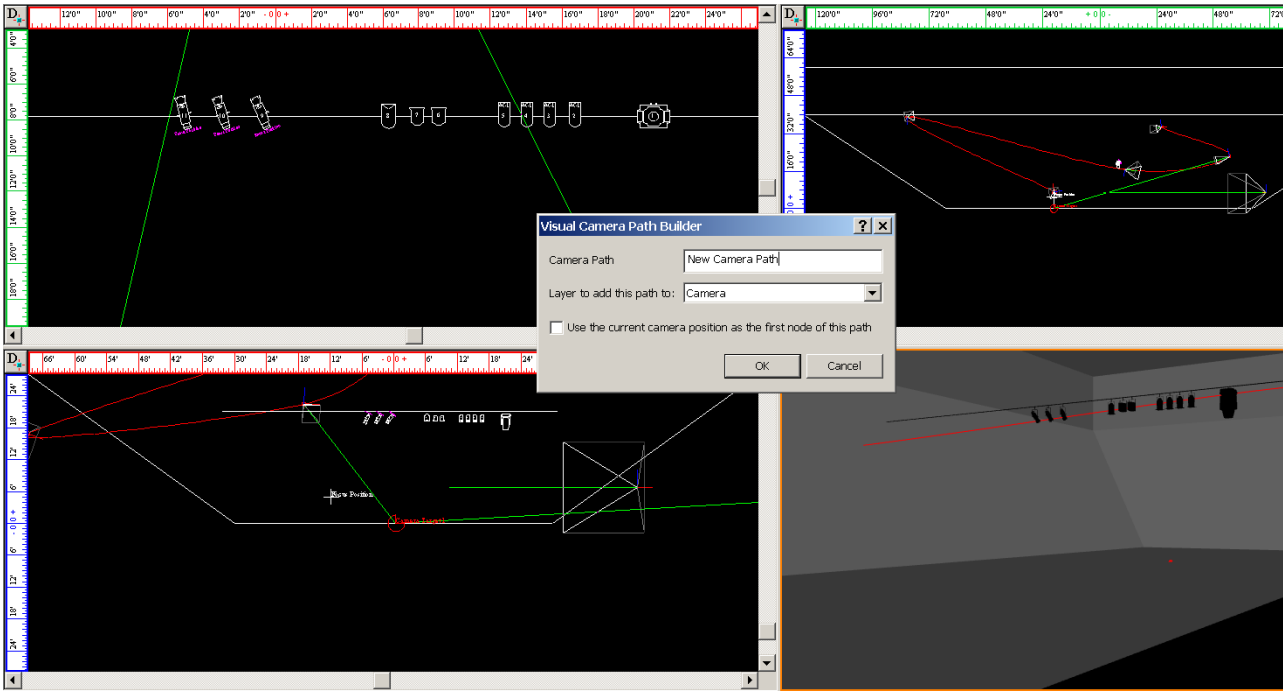
1. In Wireframe view, from the **Design** menu, click **Camera Path Editor**.
2. In the Camera Path Editor, from the Camera Path drop-down, select the Camera path that you want to edit.
3. In the Interval column, adjust the timing for each node along the path, specifying when you want the camera to be at that node.
   - If you have placed targets along the path, and you want the Camera to follow a target at a particular node, in the Target column click in the cell corresponding to the node and, from the drop-down list that appears, select *Follow Target*. Then, click in the adjacent Target cell to select the target.
   - To define the orientation and field angle of the Camera at a particular node along the path, in the Camera Target column, click in the cell corresponding the node, and then select *User Defined*.
4. Once you have defined the time intervals, you can use the Timeline tool to watch the Camera path play in the Shaded View. For details, see below.

Visual Camera Path Builder tool

The Visual Camera Path Builder tool allows you to create a Camera Path on the fly while you move the Camera around your space. This tool essentially works by placing a node at the Camera's current position once you've told WYSIWYG that you are satisfied with the Camera's current placement on screen.

To use the Visual Camera Path Builder tool

1. Right-click in the Shaded View and select **Visual Camera Path Builder**.
2. Once you've started the Visual Camera Path Builder, you'll be prompted to name your new path, add it to the Camera layer (or a layer of your choosing), and create the first node based on the Camera's current position.



3 Once you click OK in the dialog box, you will then be in Visual Camera Path Editor mode. Maneuver the Camera around the space, and when you want to create a new node based on your Camera's position, right-click in the Shaded View and choose the **Add Node**. At this point, you can also Abort your Camera Path in the Visual Camera Path Builder tool.

4 Move the Camera and right-click every time you would like to create a new node from your Camera's current position in space.

5 When you have created as many nodes as you would like to have in your Camera path, right-click again in the Shaded view and choose **Finish Camera Path**.

6 At this point, you can also Abort your Camera Path, or you can choose to Close the Camera path from your current view, which will create a seamless loop on the Camera path. When you are finished with the Visual Camera Path Builder tool, you will automatically exit the Visual Camera Path Builder tool mode.

To use the Timeline tool

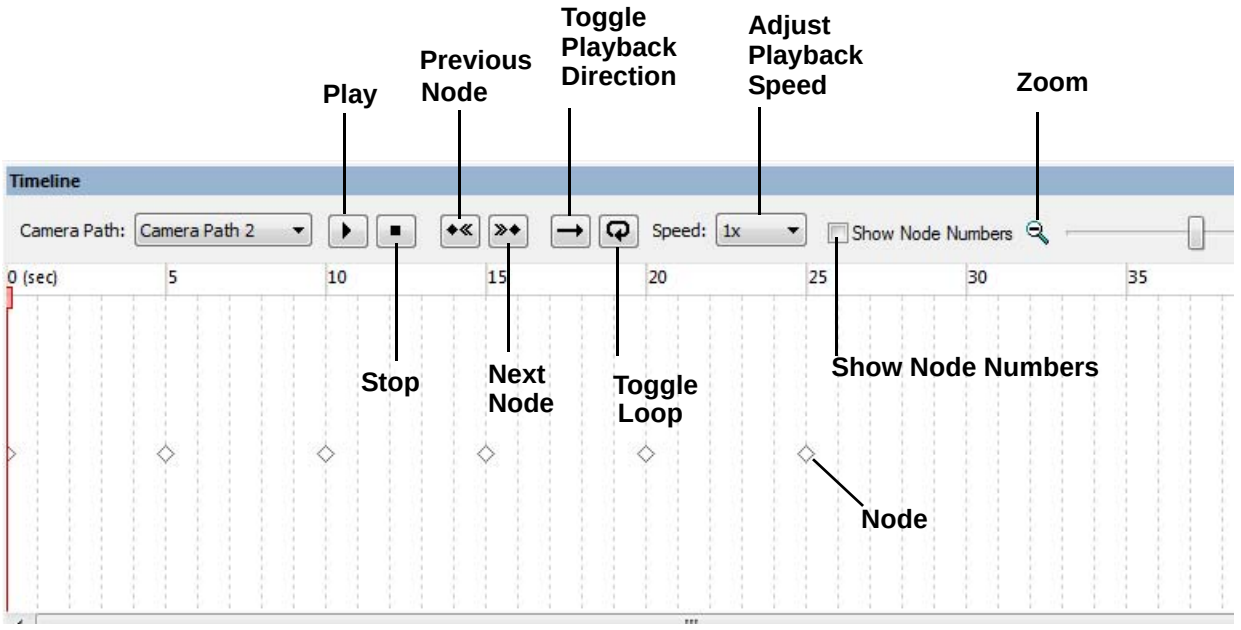
Once you have adjusted the timing of a Camera path with the Camera path editor, switch to the Shaded view to watch the camera move along the path with the Timeline tool. This tool enables you to watch the timing in slow motion, regular speed, or up to four times the speed.

Notes:

- Camera Path playback is also simulated in Full Screen mode.
- Nodes of existing Camera Paths can be edited from a Shaded view when the Timeline's Previous Node or Next Node buttons are used; they can only be edited while playback is paused.

1 In Shaded view, click **Design** > **Timeline**.

Result: The Timeline tool appears.



2 Use the controls to watch the Camera move along the path you have defined:

- **Play**: Click to start the camera path simulation in the Shaded view. While this plays, the red line moves along the timeline to display its current position. You can click on the red line indicator and scrub along the timeline, which updates the red line and the Camera simulation at the corresponding point along the path.
- **Stop**: Click to stop the camera.
- **Previous Node**: Click to switch back to the previous node on the path.
- **Next Node**: Click to move to the next node on the path.
- **Toggle PlayBack Direction**: Click to switch the direction of the camera on the path.
- **Toggle Loop**: Click to keep the camera moving on the path in a continuos loop.
- **Adjust Playback Speed**: Click to choose the playback speed.
- **Show Node Numbers**: Click to show the node numbers in the timeline.
- **Zoom**: Use the slider to zoom in or out on the timeline.

Rendering

Introduction

In WYSIWYG, you can render lighting looks to produce high quality images of your set and lighting. Rendering uses a full range of effects, including depth of field, motion blur, soft shadows, and antialiasing settings to produce a photorealistic image.

Based on the product level that you are running, when you are creating a rendering, you have two choices:

- **Render Wizard** (all product levels except Report) You can use the interactive editing and advanced visual effects of the Render Wizard to set up the look that you want to render, and then create the rendering. In this case, you must leave WYSIWYG running while the Render Wizard finishes the image. You have the choice of saving the image to an external folder that you specify or within WYSIWYG on the *Images* tab.
- **Background Rendering Manager** (all product levels except Report and Perform - Console Edition) You can use the Render Wizard to set up the look that you want to render, and then send the render job to the Background Rendering Manager to create the rendering.

  This feature lets you use the Windows taskbar to queue render jobs that will execute in the background without requiring WYSIWYG to run, letting you do other tasks while the Render Engine works independently. You can also pause and resume renderings without losing your render, and shut down/restart your computer without losing your rendering progress. For details on this feature, see “Background Rendering Manager” on page 508.

To render a lighting look

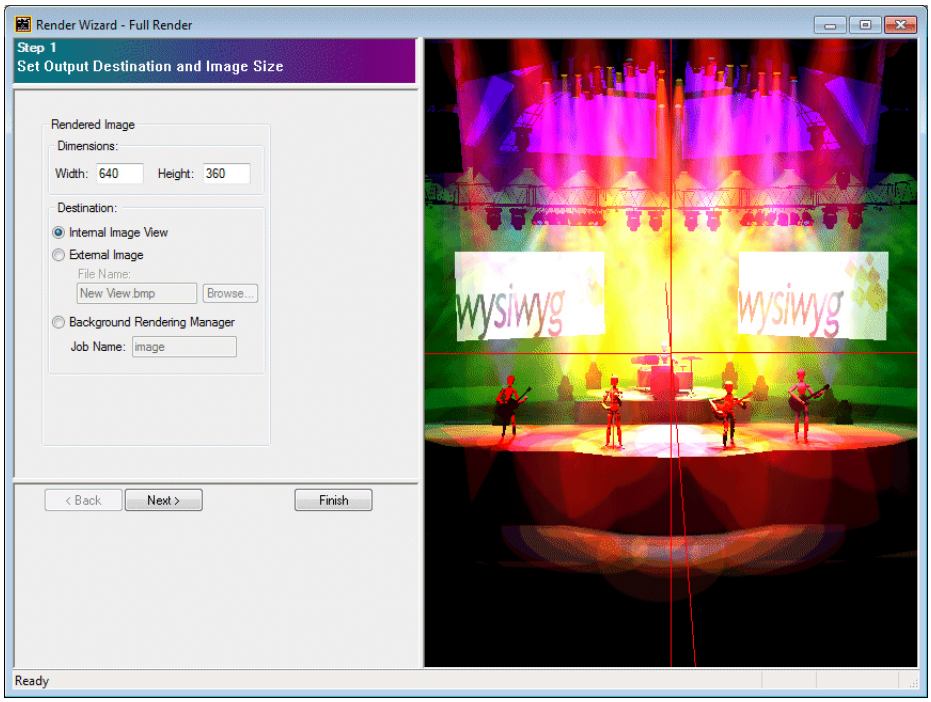
1 In the working area of the screen, display the lighting look that you want to render.

2 From the **Design** menu, choose **Render**.

Tip: You can also click the *Render Wizard* icon in the *Design* toolbar. 

Result: The Render Wizard opens and guides you through a series of options.

Step 1 - Set Output Destination and Image Size



Click in the right pane to arrange the image that you want to render. You can click and drag the image around, zoom in and out with the mouse wheel (or PAGE UP / PAGE DOWN keys), or set up your shot accurately using a camera.

Note: Make sure you use the appropriate aspect ratio (height vs. width) for the output device that you will be using to present the pictures. For example, TV is 4x3 (normally 640x480), HDTV is 16x9.

a. *Dimensions*: Type a value in pixels for the width and height of the final image.

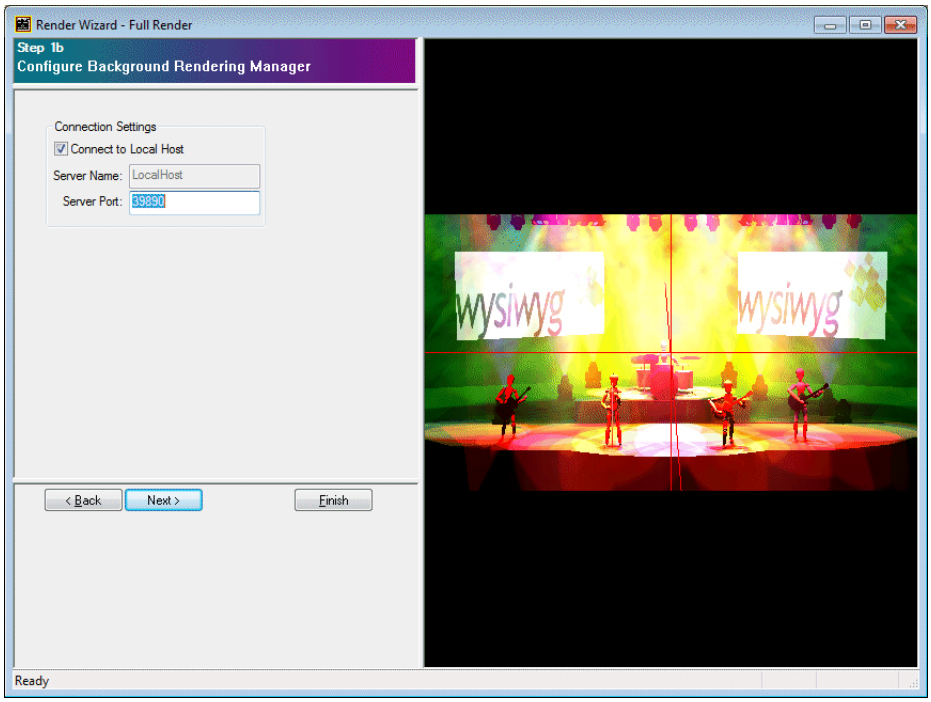
b. *Destination*: Choose where you want to save the final image and whether you want to use the Background Rendering Manager to create it.

- *Internal Image View*: Select this option to have the Render Wizard create the image and store it on the *Images* tab in Presentation mode.
- *External Image*: Select this option to have the Render Wizard create the image and store it in the location that you specify. Then type the file name for the image and click *Browse* to choose the folder where it will be saved. If desired, choose the file type, either .bmp or .jpg. If you do not make a selection, the image is stored as a .bmp image.
- *Background Rendering Manager*: Choose this option to send the render job to the Background Rendering Manager when you finish setting up your image in the Render Wizard. Type the file name for the image, ensuring that it is a unique name, and then click *Next*. For details, see “Background Rendering Manager” on page 508.

Note: If this is the first time you are using the Background Rendering Manager, you must configure it before the rendering process will begin. Upon clicking *Finish* in the Render Wizard, the Configuration Wizard appears, enabling you to configure settings such as the Network Port, the default folder for storing rendering files, and other Manager settings. For details, see “To configure the Background Rendering Manager” on page 509.

Step 1b - Configure Background Rendering Manager

This step appears only if you chose to create and save the rendering with the Background Rendering Manager. If you chose to save the image internally or externally, proceed directly to **Step 2 - Set Camera and Simulation Type**.

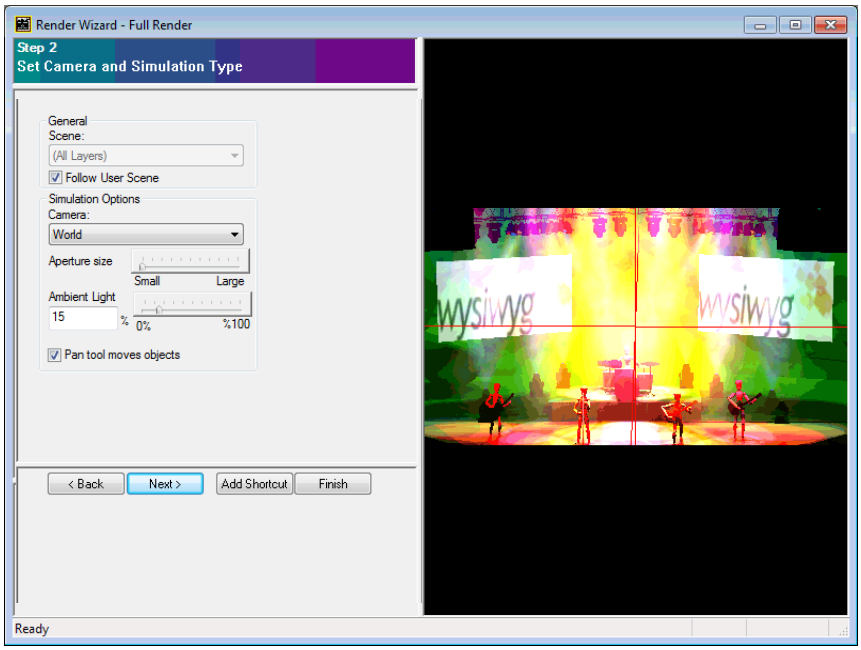


a. *Connect to Local Host*: Leave this check box selected to use the default values and have the Render Wizard send all jobs to the Background Rendering Manager that is running on your computer. To send render jobs to another computer in your network that is running the Background Rendering Manager, deselect this check box and then type the server name and port number in the boxes provided. Note that when you send render jobs to a network computer, the Background Rendering Manager must be open and running on this destination computer; in this case, the act of sending a job from the Render Wizard does not launch the Background Rendering Manager.

b. *Server Name*: Type the name of the server in your network that is running the Background Rendering Manager where you want the Render Wizard to send all render jobs. The server must be running WYSIWYG Release 18 (or higher) and have a dongle attached.

c. *Server Port*: Type the port number that the WYSIWYG Render Wizard will use to send render jobs to the Background Rendering Manager, or accept the default port number shown. It must be a value between 1025 and 65,535. For proper communication, this number must match the port number that is configured in the Background Rendering Manager on the destination computer; therefore, if you change this value, you must also change it in the Background Rendering Manager on the destination computer. For details, see “To configure the Background Rendering Manager” on page 509.

Step 2 - Set Camera and Simulation Type



Options in Step 2 affect the composition of your image.

a. *Scene*: Select a scene for the rendering. Scenes are groupings of layers used to organize a plot. For more information on scenes, refer to “Scenes” on page 223.

b. *Camera*: Select the camera for the rendering. Cameras are set up in CAD mode. For more information on cameras, refer to “Drawing cameras” on page 164.

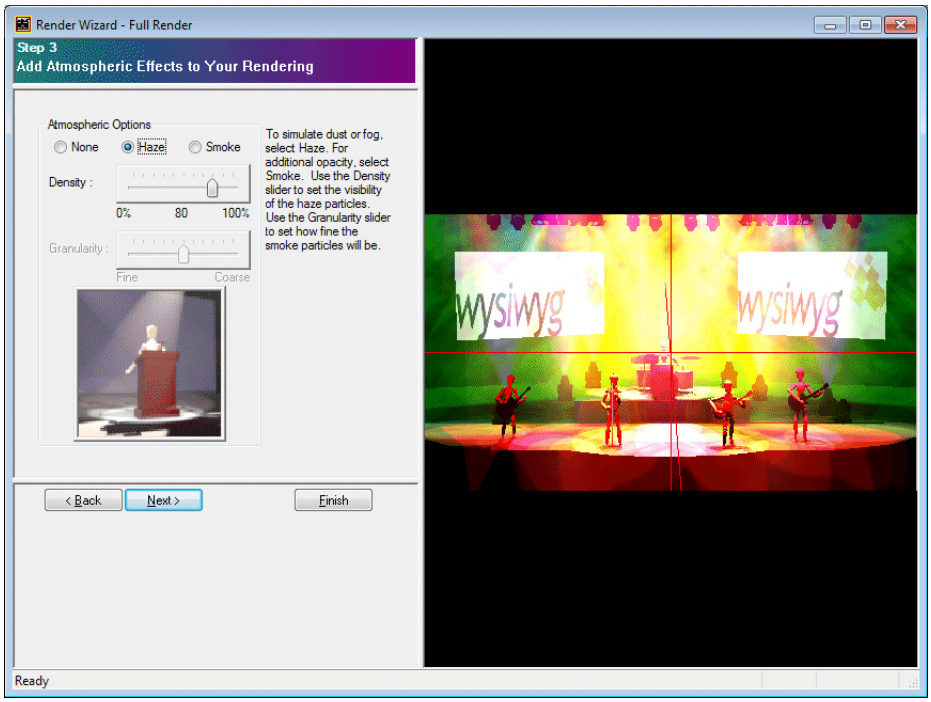
If you select a camera, you can adjust its aperture. This adjustment changes the depth of field of the final image. The larger the aperture, the “fuzzier” the rendering. The focus point is always the target of the camera.

Camera target: By default, the red lines of the target are visible. This is to aid in the composition of the image. It will not, however, be visible in your final rendering. To turn the target off (that is, set to not visible), right-click in the preview pane, and then choose *View Options*. For more information on shaded views, refer to “Modifying shaded views” on page 81.

c. *Ambient Light*: Use the slider to set the ambient light level. This adjusts the overall light level, where 0% is complete black and 100% is a bright room. This does not affect the intensity of the fixtures in your lighting look.

d. *Pan tool moves objects*: The Pan tool does not affect the outcome of your rendering. Rather the tool rotates the model on the target (if checked) or rotates the camera around the target (if unchecked). Both settings help you to compose your image.

Step 3 - Add Atmospheric Effects to Your Rendering



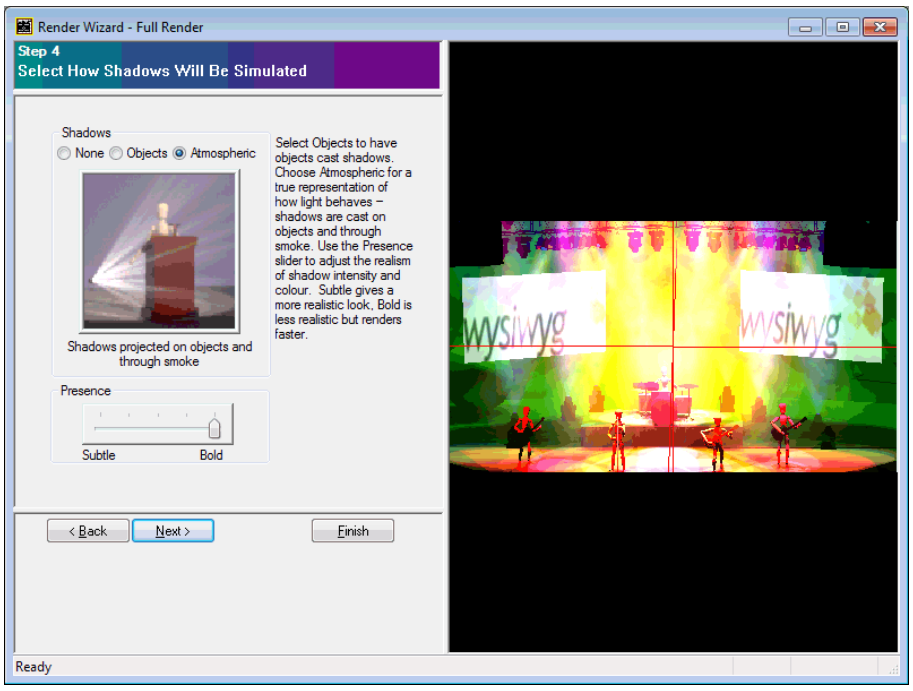
Options in Step 3 affect the atmospheric conditions in your rendering.

a. *Atmospheric Options*: The options that you set here affect the atmospheric conditions that are visible in your final rendering. To simulate atmospheric dust or fog, select *Haze*. For additional opacity, select *Smoke*.

b. *Density*: Use the slider to set the visibility of the haze particles. A setting of 0% indicates that the particles cannot be seen.

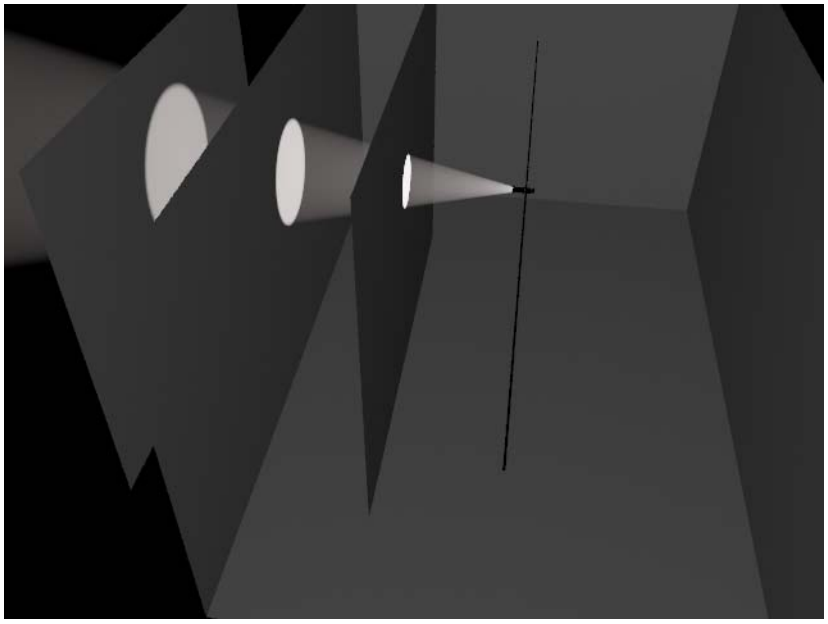
c. *Granularity*: Use the slider to set the granularity of the smoke. This setting affects how fine smoke particles are. This option is available only if you choose *Smoke* under Atmospheric Options.

Step 4 - Select How Shadows Will Be Simulated

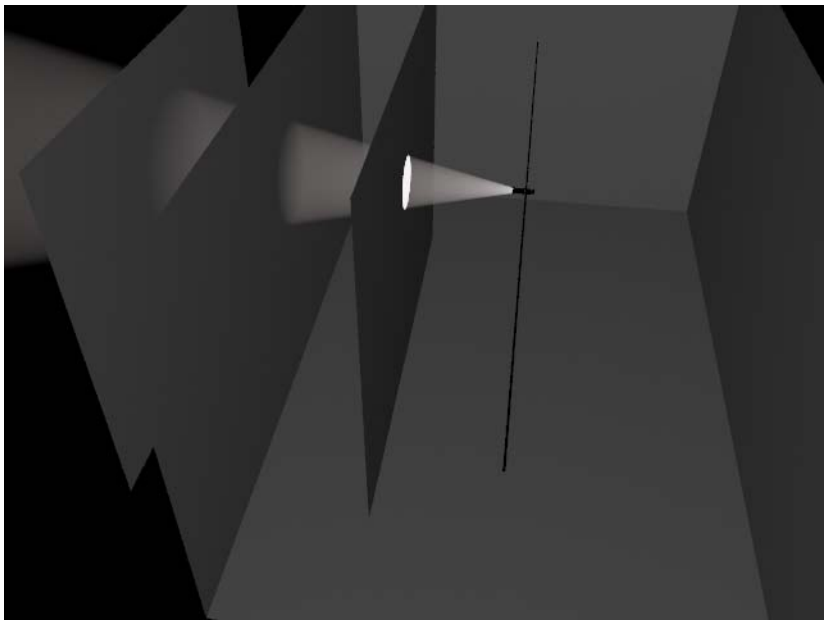


Options in Step 4 affect the projection of shadows in your rendering.

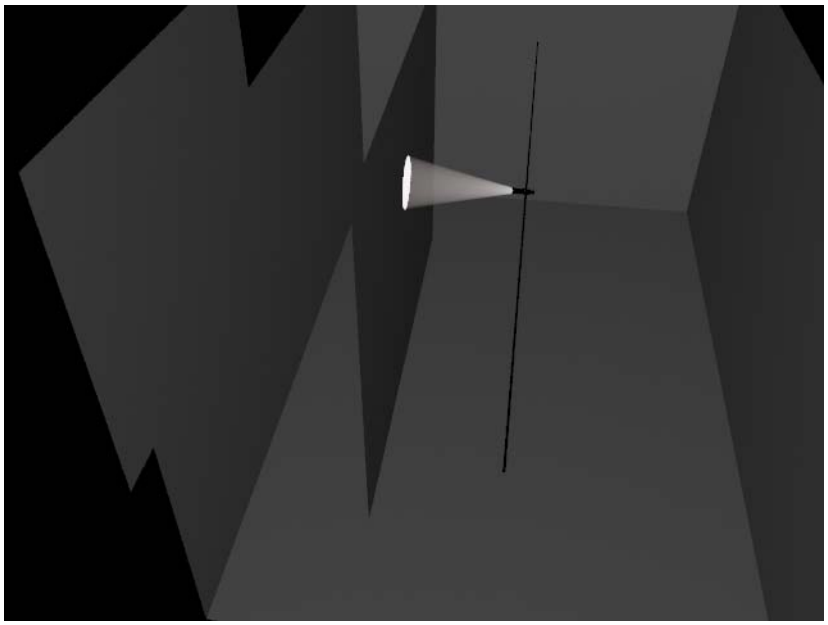
a. *Shadows*: Select how you want shadows to be projected in the final rendering. If you choose *None*, then no shadows are calculated by WYSIWYG; that is, objects in the path of the light beam do not create shadows. Instead, all objects in the path of the virtual beam are illuminated, even if an object would normally block the beam from reaching another. As illustrated in the rendering below, all three surfaces are illuminated and the beam continues into infinity.



To calculate which objects the beam of light will hit and which objects it will not hit, select *Objects*. Objects in the path of the virtual beam will not be illuminated by the beam if the beam has already been blocked by another object. However, WYSIWYG still does not calculate where the beam stops. As illustrated in the rendering below, the first surface is illuminated, but the beam is still continuing into infinity.

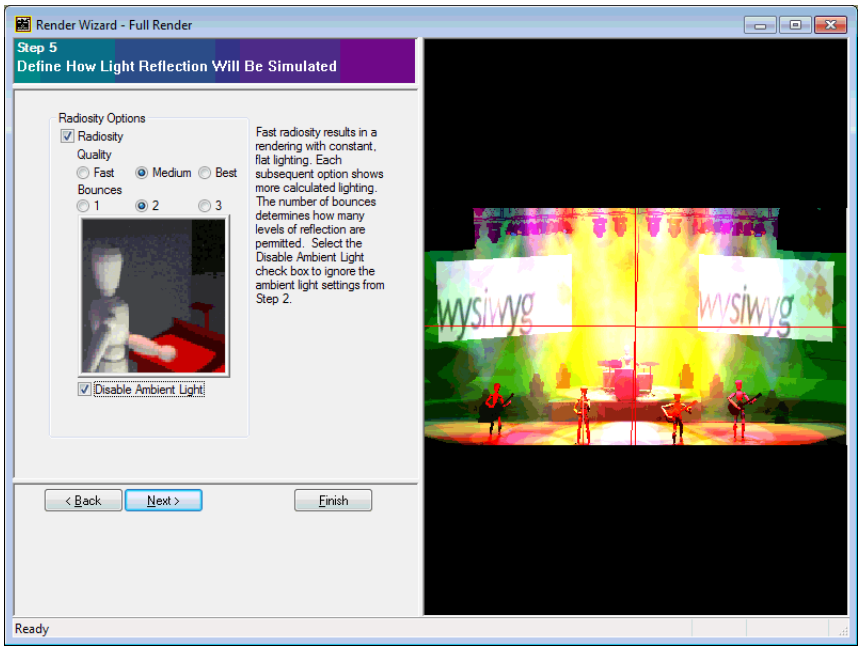


Select *Atmospheric* to calculate which objects the beam of light will hit and where the beam of light will end. This is a true representation of how light behaves. As illustrated in the rendering below, the first surface is illuminated and the beam also stops there.



b. *Presence*: Use the Presence slider to change the relative brightness of fixture beams. When set to *Subtle*, WYSIWYG accounts for all variations in lamp output, including gel transmission, color temperature, fixture efficiency, and lamp. When set to *Bold*, all fixture beams are rendered at the same general intensity.

Step 5 - Define How Light Reflection Will Be Simulated

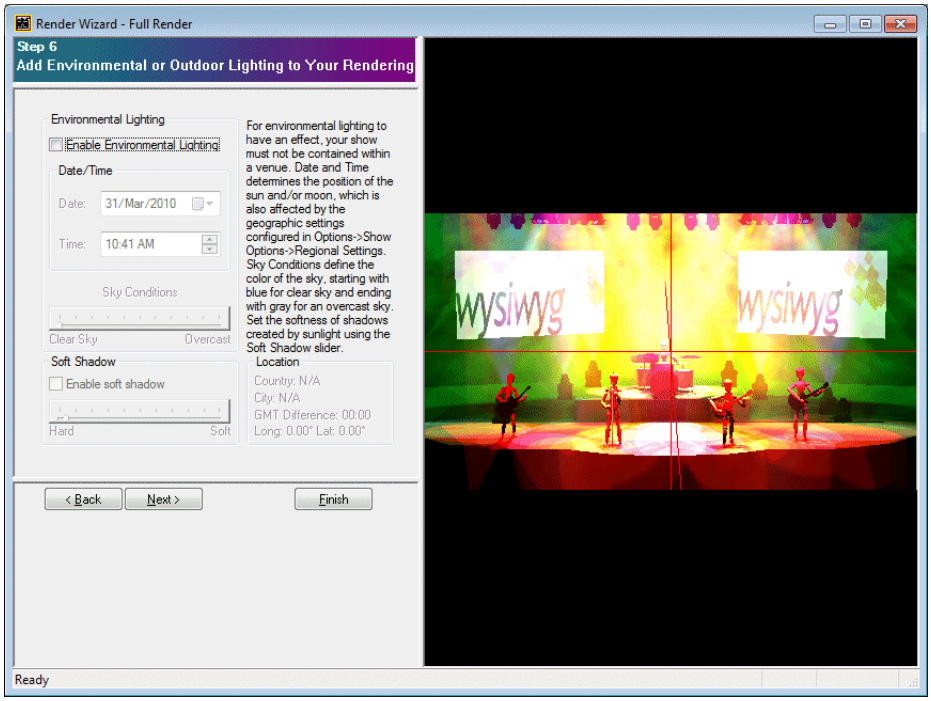


Options in Step 5 affect the radiosity level. Radiosity is defined as the combined processes of emission, transmission, and absorption of rays or reflecting beams of light. When using Radiosity, it is not necessary to use ambient lighting.

a. *Radiosity*: To generate a rendering that uses radiosity, select the *Radiosity* check box, and then choose from the available radiosity options. *Fast* radiosity results in a rendering with constant, flat lighting. Each subsequent option shows more calculated lighting. The number of bounces determines how many levels of reflection are permitted.

b. *Disable Ambient Light*: Select the *Disable Ambient Light* check box to ignore the ambient light settings from step 1.

Step 6 - Add Environmental or Outdoor Lighting to Your Rendering



Options in Step 6 affect the presence and quality of environmental or outdoor lighting. Note that for environmental lighting to have an effect, your show must not be contained within a venue.

a. *Environmental Lighting*: To generate a rendering that considers outdoor lighting conditions, select the *Environmental Lighting* check box.

b. *Date* and *Time*: The date and time of the rendering determines the position of the sun and/or moon and the resulting amount of light available. The position of the sun is also determined by the geographic settings that are configured in Show Options and the north direction that is set in CAD mode. For more information on regional settings, refer to “Regional settings tab” on page 401. For more information on setting the direction that is north, refer to “Drawing a compass” on page 170.

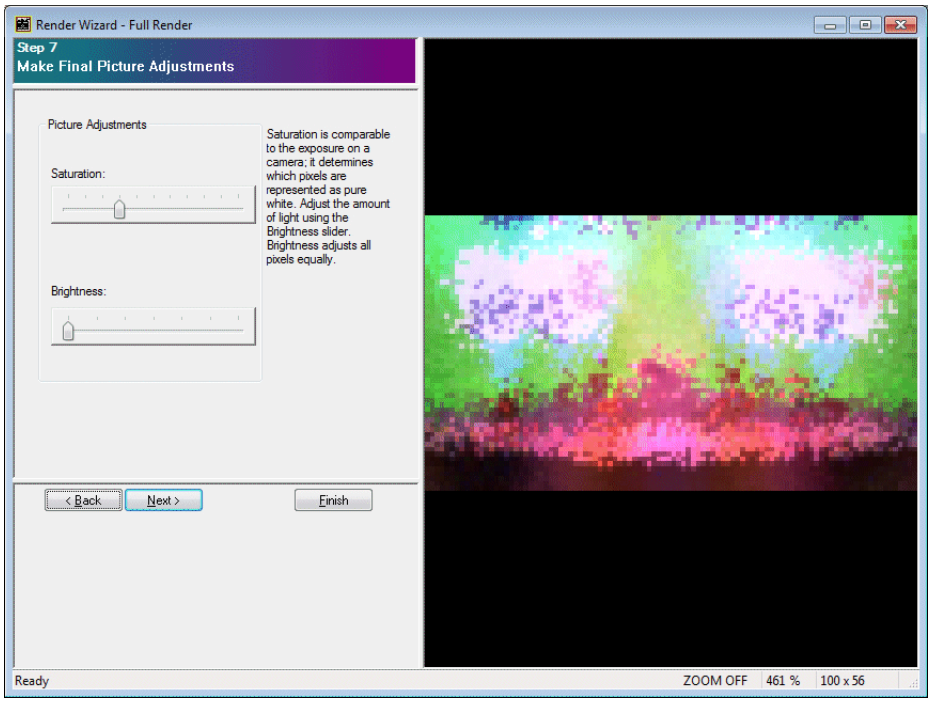
c. *Light Quality*: The amount of light available determines the brightness of the resulting rendering. Use the *Light Quality* slider to adjust the quality of light.

d. *Soft Shadow*: Click the *Enable soft shadow* check box, and then use the slider to set the softness of shadows that are created by sunlight. The softer the shadow, the less sharp the resulting shadow.

PREVIEW

When you click *Next* in Step 6, the Render Wizard calculates a small preview of the rendering and shows it in the right-hand view. It will always zoom to fit the full size of the rendered view first, but you can click in the view and zoom in and out using the PAGE UP/PAGE DOWN keys. Use the preview to fix any glaring errors in your rendering before you continue on to the final rendering.

Step 7 - Make Final Picture Adjustments

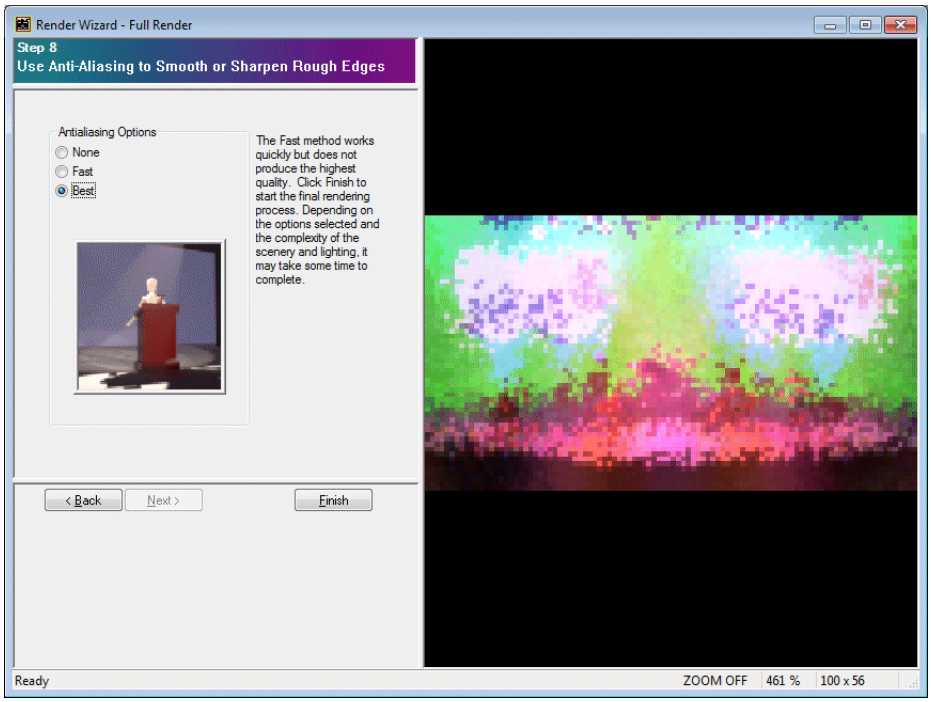


Options in Step 7 affect the quality of your final image.

a. *Saturation*: This is comparable to the exposure on a camera; it determines which pixels are represented as pure white.

b. *Brightness*: This slider adjusts all pixels equally.

Step 8 - Use Anti-Aliasing to Smooth or Sharpen Rough Edges



Options in Step 8 affect the antialiasing settings. Antialiasing is a method of smoothing out and sharpening rough or jagged edges of images to produce a more polished result. This method subsamples the pixel area and averages the results of neighboring samples to reduce the areas of high contrast (edges).

a. *Antialiasing Options*: Choose how you want edges to appear in your final rendering. The *Fast* method samples the pixels quickly, but does not produce the highest quality.

When you click *Finish*, the real rendering takes place. The total rendering time depends on the options that you selected and the complexity of the scenery and lighting. Based on the location where you chose to save the completed rendering in Step 1, your options vary upon clicking *Finish*:

- **Internal and External Images** If you chose to save the rendering internally or externally, then you can minimize the Render Wizard window and continue working on other parts of your document as it renders. You can use the arrow keys, PAGE UP/PAGE DOWN, mouse or scroll bars to explore the picture as it is being produced. The final image is saved to the destination you specified in step 1.
- **Background Rendering Manager Images** If you sent the render job to the Background Rendering Manager, then a status window appears, showing the progress as the render job is sent to the Background Rendering Manager on the destination computer (either your own or another server in the network).

  When the job has been sent successfully, you can open the Background Rendering Manager to check its progress. You can close the Render Wizard and WYSIWYG, queue jobs, pause and resume them, and shut down/restart your computer without losing the rendering progress. For details, see “To use the Background Rendering Manager” on page 512.

Note: If this is the first time you are using the Background Rendering Manager, you must first configure it before the rendering process will begin. Upon clicking *Finish* in the Render Wizard, the Configuration Wizard appears, enabling you to configure settings such as the Network Port, the default folder for storing rendering files, and other Manager settings. For details, see “To configure the Background Rendering Manager” on page 509.

Saving render settings

You can save the options that you set in the Render Wizard as a shortcut for quick access to these settings at a future time. There are two ways of saving a Render shortcut:

- from the Render Wizard
- from the Render shortcut bar, in a similar manner as other shortcuts

The Render shortcut bar is prepopulated with two shortcuts: Quick Render and Full Render. Quick Render shortcuts provide settings for a small, lower-quality render, while Full Render shortcuts provide settings for a larger, high-quality rendering.

To create a shortcut from the Render Wizard

1 From the **Design** menu, choose **Render**.

2 While working through the steps in the Render Wizard, you can, at any time, click *Add Shortcut* to save the current settings as a shortcut.

3 Type a name for the shortcut.

4 Click *OK*.

Result: The shortcut is added to the Render shortcut bar.

To create a shortcut from the Render shortcut bar

1 Right-click in the open space on the Render shortcut bar, and then choose **New Render**.

2 Type a name for the new render.

3 Click *OK*.

Result: The Render shortcut is saved on the Render shortcut bar. The shortcut is preconfigured with the default values for each of the eight steps. These defaults are the same values that are used when you choose **Render** from the menu bar or the toolbar.

To modify a Render shortcut

1 Double-click on the Render shortcut that you want to modify.

Result: The Render Wizard opens, with the values that are saved in the shortcut.

2 Modify the options as required.

Result: The settings are recorded in the shortcut and are available the next time that you select the shortcut.

Background Rendering Manager

Introduction

Once you have used the Render Wizard to set up the look that you want to render, you can send the render job to the Background Rendering Manager to create the actual rendering. You can send multiple jobs to the Background Rendering Manager queue and then arrange the order of the queue or remove jobs from it. While you can send multiple jobs, the Background Rendering Manager processes only one job at a time, the job at the top of the queue. Once this job is finished, it will then automatically proceed to the next queued job, and so on.

If you manually move a job in progress from the top of the queue down, you pause the rendering and start processing the next job in the queue. However, you do not lose the progress of any rendering jobs that have been started. When they reach the top of the queue, the Background Rendering Manager resumes rendering the image where it last left off.

The Background Rendering Manager processes render jobs independently without requiring WYSIWYG to run. This feature lets you work on other tasks while the renderings in the queue are processed in the order you arranged. You can work on new WYSIWYG files, or you can close WYSIWYG altogether and perform other tasks. You can even shut down your computer and restart later without losing the rendering progress.

When setting up renderings in the Render Wizard, you can send jobs to the Background Rendering Manager that resides on the same computer, or to another computer in your network that is also running the Background Rendering Manager (and has a dongle attached). By sending all rendering jobs to another computer, you free up CPU capacity on your own computer.

Configuring the Background Rendering Manager

When you open the Background Rendering Manager for the first time, the Configuration Wizard appears, prompting you to choose settings, such as the network port, the startup options, and the folder where all renderings are stored.

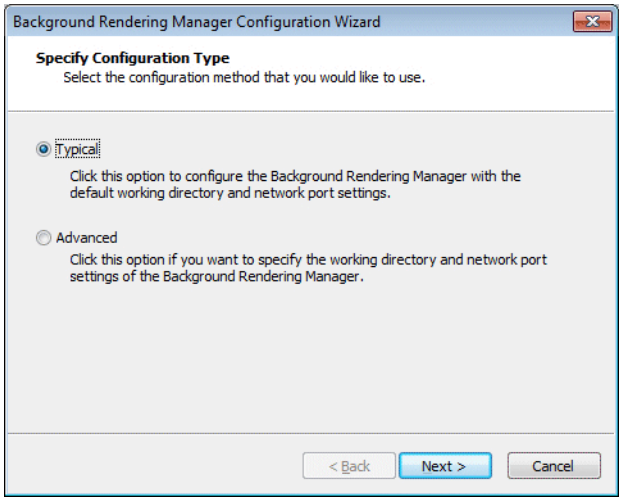
This Wizard appears automatically when:

- You click **Start** > **All Programs** > **WYSIWYG** > **WYSIWYG Background Rendering Manager** to open the program for the first time.
- You click *Finish* in the Render Wizard after setting up a rendering and choosing to send it to the Background Rendering Manager for the first time.

To configure the Background Rendering Manager

1 Click **Start** > **All Programs** > **WYSIWYG** > **WYSIWYG Background Rendering Manager**.

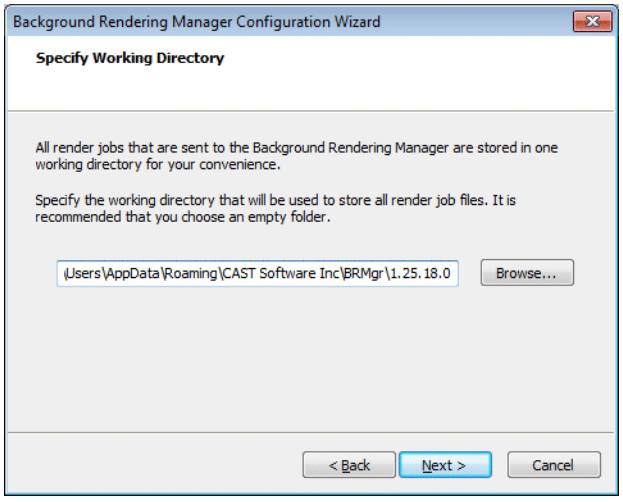
2 In the Welcome window, click *Next*.



3 Select whether you want the program to use the default settings or custom settings that you specify.

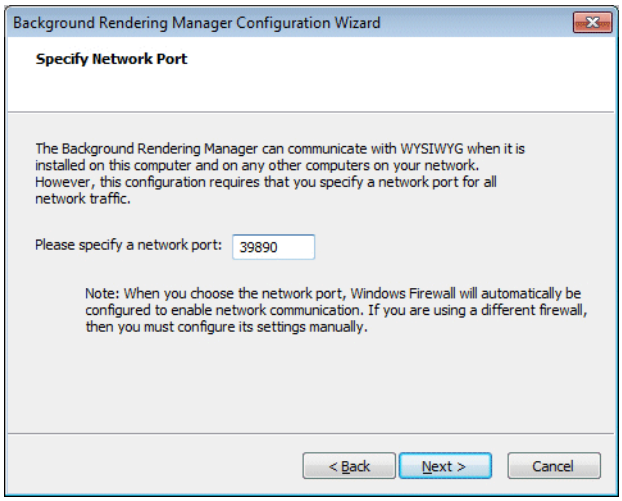
- **Typical** Click this option to configure the program with the default working directory and network port settings.
- **Advanced** Click this option to specify the working directory and network port settings for the program.

4 Click *Next*. If you chose *Typical* in the previous step, proceed directly to step 9. If you chose *Advanced*, the following window appears:



5 The working directory is used to store files that the render engine requires to generate renderings. To change the default working directory shown (for example, if you have run out of space on the default drive), type the path and new folder name in the box provided, or click *Browse* to locate the folder. It is recommended that the folder be new and empty.

6 Click *Next*.

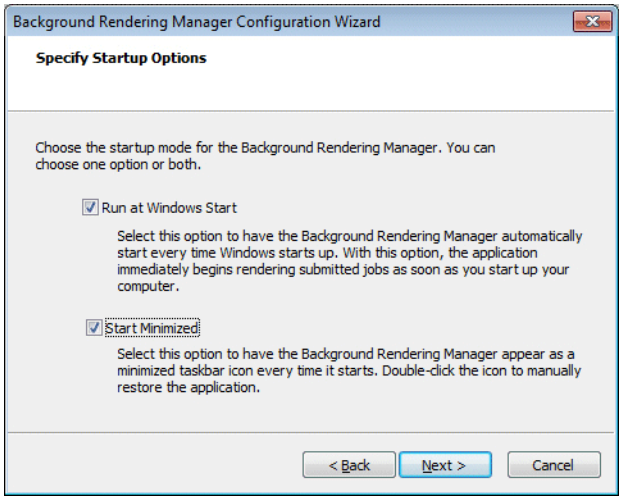


7 Type the incoming Network Port that the Background Rendering Manager will monitor for rendering jobs sent from WYSIWYG.

Notes:

- If you change this default port, you must also change the outbound port configured in the Render Wizard for sending jobs to this Background Rendering Manager. These two ports must match.
- When you choose the network port, Windows Firewall will automatically be configured to enable network communication. If you are using a different firewall, then you must configure its settings manually.

8 Click *Next*.



9 Choose the startup mode for the Background Rendering Manager. You can choose one option or both.

- **Run at Windows Startup** Select this option to have the Background Rendering Manager automatically start every time Windows starts up. With this option, the application immediately begins rendering submitted jobs as soon as you start up your computer, unless it was paused on last exit.
- **Start Minimized** Select this option to have the Background Rendering Manager appear as a minimized taskbar icon every time it starts. Double-click the icon to manually restore the application to its full size. If you select *only* this option, the Background Rendering Manager starts when you send a render job to it from a Render Wizard that is installed on this same computer. However, if you are sending render jobs from a different computer, then you

will have to manually open the Background Rendering Manager on this computer before it will accept render jobs.

10 Click *Next*.

11 Click *Finish*.

Result: The Background Rendering Manager is now ready to use.

Tip: To change these settings at any time, you can run the Configuration Wizard again by clicking **Tools** > **Configuration Wizard**. You can also click **Tools** > **Options** to change these settings on one window.

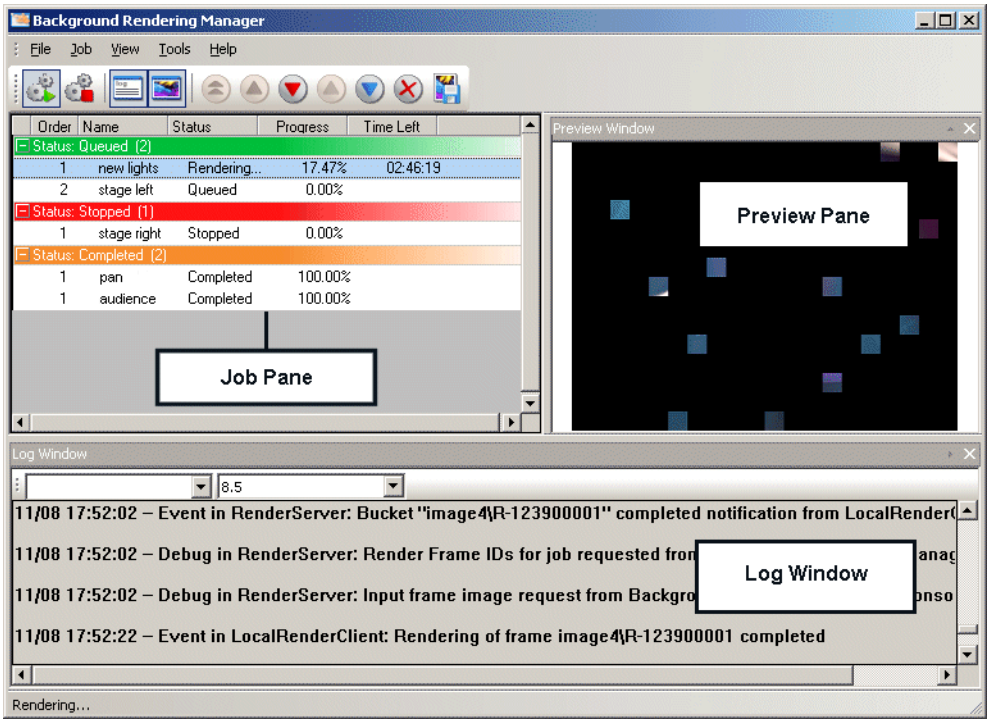
Background Rendering Manager taskbar icons

Based on the settings that you chose when configuring the Background Rendering Manager, when the program is started, it appears either as a maximized window or a minimized icon on the right side of the Windows taskbar. The icon changes according to the current state of the program. Double-click the taskbar icon to open the program.

Icon	Status
	When this icon appears, the program is idle. Either it has not been configured yet, or no jobs have been sent to it.
	When this icon appears, the program is currently processing a job in the queue.
	When this icon appears, the program is paused.
	When this icon appears, the program has completed all jobs in the queue.

Background Rendering Manager window

When you open the Background Rendering Manager (either by sending a render job to it, or double-clicking the taskbar icon), the main window appears, showing you the status of all rendering jobs.



Job pane

The Job pane shows you all jobs that have been successfully sent to the Background Rendering Manager. While you can send multiple jobs, the Background Rendering Manager processes only one job at a time, the job at the top of the queue. Once this job is finished, it will then automatically proceed to the next queued job, and so on. For tips on managing the jobs listed, see “Managing jobs in the Background Rendering Manager queue” on page 513.

Preview pane

This window shows you a visual representation of the rendering progress. Note that the final image is scaled to fit this pane and is not indicative of the quality of the actual rendering. To view the final rendering, you must export the image as a graphic file. For details, see “To export rendered images from the Background Rendering Manager” on page 515.

Log window

Check the messages in the log window for an up-to-date status as the rendering job is processed. You can change and resize the font in this window by using the drop-down boxes provided.

Using the Background Rendering Manager

Once you have configured the Background Rendering Manager, it is ready to accept render jobs from WYSIWYG.

To use the Background Rendering Manager

1. In WYSIWYG, open the Render Wizard and set up the rendering that you want to create, choosing *Background Rendering Manager* in Step 1.
2. Proceed through the steps of the Render Wizard, and then click *Finish*.

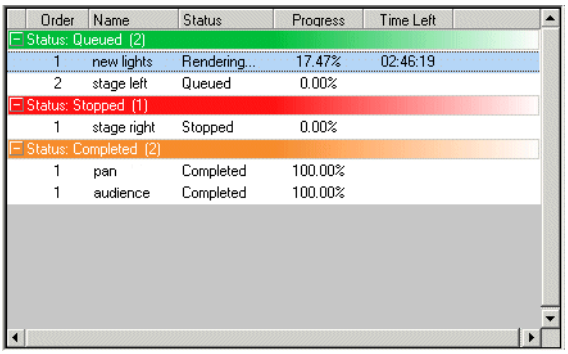
   Result: The *Submitting Render Job* window appears, showing the progress as the job is sent to the Background Rendering Manager.
3. When the job is successfully sent, a WYSIWYG message box notifies you. The next step differs based on where you sent the job (either to your own computer, or another computer on the network):
   - **Background Rendering Manager residing on your computer** If you have sent the job to the Background Rendering Manager residing on the same computer as the Render Wizard, it automatically starts processing the rendering if there are no jobs currently being rendered (if it is currently processing a job, the new job is placed in the active queue).
   - **Background Rendering Manager residing on a network computer** If you have sent the job to the Background Rendering Manager residing a network computer, you must first ensure that the Background Rendering Manager is started before it will process the job. The act of sending a rendering job does not start the Background Rendering Manager when it is installed on a network computer.
4. Open the Background Rendering Manager to check the progress of the rendering and arrange the rendering queue. For details, see “Managing jobs in the Background Rendering Manager queue” on page 513.

   Note: You can shut down your computer while the Background Rendering Manager is processing the job without losing the rendering progress. You can also pause and resume the program. When you restart your computer or resume the program, the rendering continues where it last left off. (You might need to open the Background Rendering Manager manually before it will resume rendering.)

Managing jobs in the Background Rendering Manager queue

The Job pane shows you all jobs that have been successfully sent to the Background Rendering Manager. While you can send multiple jobs, the Background Rendering Manager processes only one job at a time, the job at the top of the queue. Once this job is finished, it will then automatically proceed to the next queued job, and so on.

As shown in the Status column in the following graphic, rendering jobs can have one of four statuses:



- **Rendering** This is the job that is currently being processed. Only one job can have this status at any given time. The values in the *Progress* and *Time Left* columns indicate when the job will be finished.
- **Queued** Jobs with this status are waiting to be processed. These can be new jobs or those that have already started, but have been interrupted by moving them down in the queue. In this latter case, when the job is moved back to the top of the queue, the rendering automatically resumes where it left off.
- **Stopped** Jobs with this status have been manually dequeued. This means that they will not automatically move up in the queue as jobs are completed. To process jobs with this status, you have to highlight them and click *Queue* to move them back into the active queue (or click *Render Job Now* to start rendering the job immediately).
- **Completed** These are jobs that have been processed successfully. However, the actual rendering image still resides in the Background Rendering Manager. To export the image to a folder of your choice, highlight the completed job, and then click *Save Rendering*.

Background Rendering Manager buttons

The table below contains descriptions of the Background Rendering Manager buttons. Use these descriptions as a guideline when managing the jobs in the queue.

Button	Explanation
	**Resume Rendering**: This button is enabled when you have paused all jobs in the Background Rendering Manager. Click this button to resume the rendering process, starting with the first job in the queue. The rendering continues where it last left off.
	**Pause Rendering**: This button is enabled when the Background Rendering Manager is currently processing a render job. Click this button to pause all rendering activity in the Background Rendering Manager.
	**Show Log Window**: Click this button to show or hide the log window, listing all status and error messages of the Background Rendering Manager.
	**Show Render Window**: Click this button to show or hide the window that shows the progress of the rendering.

Button	Explanation
	**Render Job Now**: When you have multiple render jobs in the queue waiting to be processed, highlight any rendering job with the *Queued* status and press this button to move the job to the top of the queue. The current rendering is paused and moved down one position and the new render job starts immediately.
	**Queue Job**: This button is enabled only when you have selected a dequeued job (it is stopped, but its rendering progress is retained). Highlight the stopped job and then click this button to move it back into the queue. If there is a job being processed, the newly queued job is placed below it.
	**Dequeue Job**: Highlight a job with the Queued status and then press this button to give the job the Stopped status. You can dequeue any job in the queue, those that are waiting to be processed as well as the job that is currently being rendered. When you dequeue a job, the Background Rendering Manager retains its rendering progress and resumes the rendering when you move the job back up to the top of the queue.
	**Move Job Up**: This button enables you to move jobs up within their current status level when you have multiple jobs that are either Stopped or Queued. You cannot use this button to move a stopped job into the active queue. However, you can click the bottom job in the Queued status area and use this button to move it all the way to the top of the queue so that it starts being processed (click *Render Job Now* as a shortcut). You can also click the bottom job in the Stopped status area and move it up within this same area, if desired.
	**Move Job Down**: This button enables you to move jobs down within their current status level when you have multiple jobs that are either Stopped or Queued. You cannot use this button to move a queued job into the Stopped status. However, you can click the top job in the Queued status area (the job that is currently being processed) and use this button to move it all the way to the bottom of the queue. You can also click the top job in the Stopped status area and move it down within this same area, if desired.
	**Delete Job**: Highlight any job in the Job pane—those with the Rendering, Queued, Stopped, or Completed status—and press this button to delete the job and all supporting files from the Background Rendering Manager. Note that when you delete jobs, you cannot recover them and there will be no record of them.
	**Save Rendering**: Highlight a Completed job that you want to save, and then click this button to export the rendering as a graphic file to the folder of your choice. You are then prompted to choose the location for the file and the file type (.bmp, .jpg, .tif, or .gif).

To pause and resume all jobs in the Background Rendering Manager

To free up some CPU capacity on your computer (for example, if you are going to run another program), you can pause the Background Rendering Manager and the job that is currently in progress. When you click *Resume*, the job at the top of the queue continues rendering where it left off.

1 In the Background Rendering Manager window, click the *Pause* button. 

2 Click *Resume* when you want to resume rendering the job at the top of the queue.



To move a render job to the top of the queue

When you move a Queued job to the top of the queue, the Background Rendering Manager automatically starts to process it. The current job is moved down one step in the queue to a paused status (any progress that had been made in the rendering is saved with the job).

1 In the Background Rendering Manager window, highlight the Queued job that you want to start rendering.

2 Press the *Move Job Up* button until the job is at the top of the queue. 

Result: The job starts rendering.

Tip: If you have many jobs in the queue, click the *Render Job Now* button to move a job from the bottom of the queue to the top and start rendering it immediately.

To remove a render job from the queue

When you remove a job from the queue, it will not automatically move up in the queue as jobs are completed. Instead, to process jobs with this status, you have to highlight them and click *Queue* to move them back into the active queue.

1 In the Background Rendering Manager window, highlight the Rendering or Queued job that you want to remove from the queue.

2 Press the *Dequeue* button. 

Result: The job is moved out of the queue and given the Stopped status.

To move a stopped render job back into the queue

When you dequeue a job, you cannot use the *Move Job Up* button to move it back into the active queue; instead, you must use the *Queue* button to place the job back into the queue. The job will be processed when it reaches the top of the queue.

1 In the Background Rendering Manager window, highlight the Stopped job that you want to move back into the queue.

2 Press the *Queue* button. 

Result: The job moves to the active queue.

To move render jobs up or down in the queue

When you have multiple jobs that are either Queued or Stopped, you can move jobs up or down within their current status level. For example, you can use this feature to arrange the order in which the Background Rendering Manager will process queued jobs, or you can move a queued job to the top of the queue so that it is processed immediately.

1 In the Background Rendering Manager window, highlight the Queued or Stopped job that you want to move.

2 Press the *Move Job Up* or *Move Job Down* button repeatedly until the desired position is reached in the queue.  

To export rendered images from the Background Rendering Manager

To use the rendered images that you create with the Background Rendering Manager, you must export them to a folder of your choice, in a graphic format like .bmp, .jpg, or .gif.

1 In the Background Rendering Manager window, highlight the Completed job that you want to export.

2 Click the *Save Rendering* button. 

3 Browse to the location where you want to save the rendered image.

4 In the *File name* box, type the name of the file.

5 From the *Save as type* drop-down box, choose a graphic format (.bmp, .jpg, .tif or .gif).

6 Click *Save*.

To delete render jobs from the Background Rendering Manager

You can delete any job—those with the Rendering, Queued, Stopped, or Completed status—and all of its supporting files from the Background Rendering Manager. Note that when you delete jobs, you cannot recover them and there will be no record of them.

1 In the Background Rendering Manager window, highlight the job that you want to delete.

2 Press the *Delete* button. A message box asks you to confirm your choice. 

3 Click *Yes*.

Chapter 10

Smart Cells



Smart cells allow you to reference WYSIWYG variables in reports, worksheets and layouts to ensure that the same type of information is displayed in each. These variables are found in the report headings and in various title blocks in PRES mode.

In this chapter

Smart Cells 519



Smart Cells

Introduction

Smart cells allow you to reference WYSIWYG variables in reports and worksheets to ensure that the same type of information is displayed in each report and worksheet. These variables are found in the report headings and in various title blocks in PRES mode.

Additionally, you can replace these variables in your Layout text by typing them directly. To do so, you must surround the variable with per cent symbols (%) when altering the text item's properties. For example, to add the Chief Lighting Designer variable, you would type %Chief Lighting Designer%.

All variables shown in the Production Team Info Wizard and Production Team Info tab of the Options > Show Options window are available as smart cells. For more information on changing these variables, see "Production Team Info Wizard" on page 469.

The following variables are accessible from a Smart Cell drop-down (or by typing them in Text items in Layouts):

			Available In		
Variable	**Definition**	**Source**	**Reports**	**Worksheets**	**Layouts**
Production Team Info variables	Any Production Team variable	Any variable as defined on the Production Team Info tab in Show Options. For more information on this tab, refer to "Production Team Info tab" on page 396.	Y	Y	Y
Data	The data that comprises the report	Information that is available in Data mode	Y	N	N
Show Date	The date of the show	The date of the show defined on the Production Team Info tab in Show Options. For more information on this tab, refer to "Production Team Info tab" on page 396.	Y	Y	Y

Variable	Definition	Source	Available In		
			Reports	Worksheets	Layouts
File	The name of your WYSIWYG file	The name you chose when saving your document using either the Save or Save As command from the File menu.	Y	Y	Y
Group	The name of the group	The Group By parameter chosen on the Report tab in View options. For more information on this option, refer to “Report tab” on page 415.	Y	N	N
Headings	The column headings that comprise the report	The columns defined in the Report tab in View Options. For more information on choosing these columns, refer to “Report tab” on page 415.	Y	N	N
Scene	The scene chosen for the report	The scene defined on the General tab in View Options	Y	Y	Y
Show Name	The name of the show	The show defined on the Production Team Info tab in Show Options. For more information on this tab, refer to “Production Team Info tab” on page 396.	Y	Y	Y
Title	The title of the report	The name of the report as defined in the shortcut bar (for example, Instrument Count).	Y	Y	Y

Variable	Definition	Source	Available In		
			Reports	Worksheets	Layouts
User	The name of the current user	The user name defined in the Show Options, Document Summary tab, Last Modified By field.	Y	Y	Y
Venue	The name of the Venue	The venue defined on the Production Team Info tab in Show Options. For more information on this tab, refer to “Production Team Info tab” on page 396.	Y	Y	Y

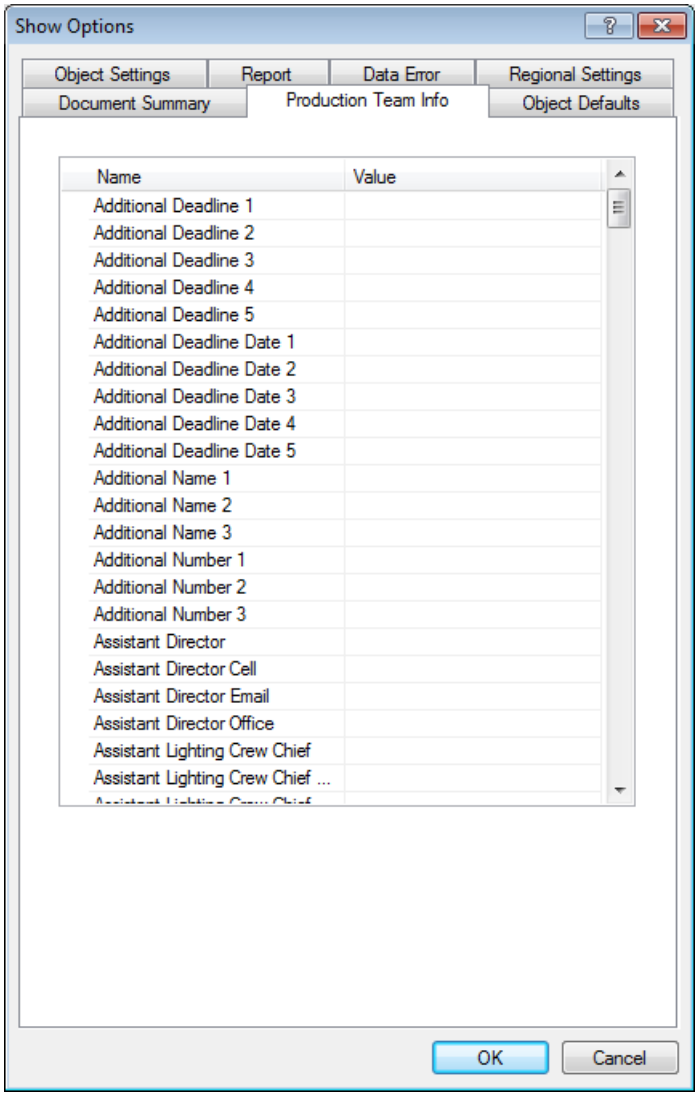
Show information

You can enter all of your show information at once using the Production Team Info Wizard. For details, see “Production Team Info Wizard” on page 469. Alternately, you can enter your information manually using the edit fields in the Production Team Info tab. These fields appear in the headings of all reports and in the WYSIWYG default title block (for plots). They also appear in Smart Cells, if you have chosen to use them.

To input show info in the Production Team Info tab

1 From the **Options** menu, choose **Show Options**.

2 Click the *Production Team Info* tab.



3 In the Value column, type all the pertinent information in the boxes provided.

4 Click *OK*.

Result: All reports are updated.

Chapter 11

Live mode



Live mode is used for graphically simulating the output of a lighting control console or compatible offline editor. Live mode also has rendering ability, enabling you to create photo-realistic pictures of the simulated lighting looks. Unlike Design mode, Live mode displays transitions from cue to cue, allowing you to see the programmed movement of light over time. If you have configured a patched video source, you can use Live mode to control the progress of your video with the console device. And if you have patched and mapped an image to a light-emitting surface, you can control it with the Image Console.

Live mode is available only in WYSIWYG Perform.

In this chapter

Layout tabs	525
Connecting to consoles	526
Working with consoles	530
Image Console	535
AutoPatch	539
Visualization	541
Autofocus	544
Rendering	547
DMX Camera	548
Patching camera paths	550



Layout tabs

Introduction

Beneath the work area in each mode is a series of layout tabs. These tabs provide various configurations of the views that you are working with. To change layouts, click the appropriate tab.

Live mode contains the following layout tabs:

- Wireframe - The work area displays a full-screen wireframe view.
- Quad - The work area is divided into quadrants, three of which can be modified to show plan, front, or side views. The lower-right quadrant contains a shaded view.
- Shaded - The work area displays a full-screen shaded view.

Note: The wireframe views in Live mode are used for fixture selection only. To make any drawing modifications, you must return to CAD mode.

Connecting to consoles

Introduction

To use Live mode, you need to insert and connect to a control console or compatible offline editor. If you use a DMX console, you require a DMX interface device to connect to WYSIWYG.

You must ensure that the patch you have created in WYSIWYG and the patch in your console are exactly the same for simulation to work properly.

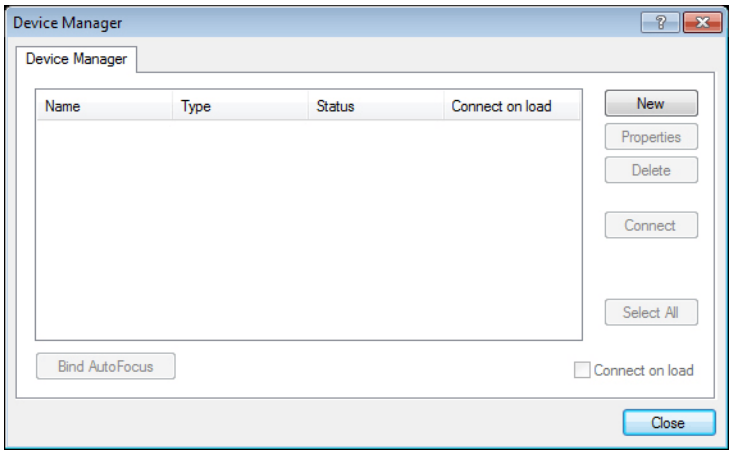
Connecting to a console

To connect to a console

1 Connect the console to the computer with the appropriate hardware.

2 From the **Live** menu, choose **Device Manager**.

Result: The *Device Manager* dialog box is displayed.



3 Click *New*.

Result: The console library is displayed.

4 Navigate through the console library until you find the console or device that you are connecting to. Click the console name to highlight it.

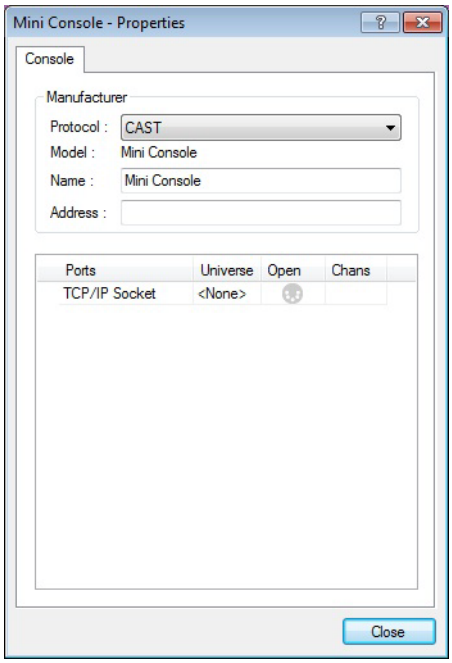
Note: If you are connecting to a console with a Wyg-it or Wyg-it2, you can select either the console or the Wyg-it device. The *Address* box discussed below must contain the name of the Wyg-it.

5 Click *Insert*.

Result: The console appears in the *Device Manager*.

6 With the console name still selected, click *Properties*.

Result: The *Properties* dialog box is displayed.



7 From the *Protocol* drop-down list, select the protocol used by the console. For example, DMX, SandNet.

Result: The console model is displayed in the *Model* box. The name of the console is displayed in the *Name* box.

8 In the *Address* box, type the designated address of the console, if applicable.

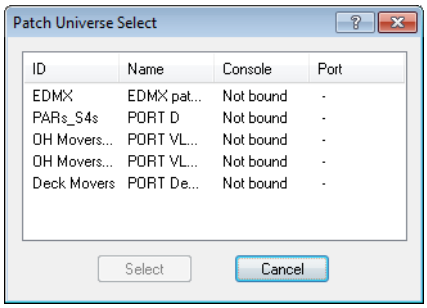
9 A list of the output ports from the console appears in the *Ports* list. You must bind these outputs to WYSIWYG patch universes. Click the appropriate port to highlight it.

Note: Motion control systems have a single port named “Motion.”

10 Click *Bind*.

Note: If you are working with a motion control system, then you can bind only to motion universes; if you are working with a DMX console then you can bind only to DMX universes.

Result: The Patch Universe Select dialog box is displayed.



11 Select the desired Patch Universe, and then click *Select*.

12 Repeat the above steps to bind all ports to their appropriate patch universes.

Note: You cannot control a channel count exceeding that of the WYSIWYG Perform package that you purchased - 500, 1000, 2000, 5000, and so on.

13 Click *OK* to close the *Properties* box.

14 Click *Connect*.

Result: The status of the console changes to “connected.”

15 Click *OK* to exit the Device Manager.

Note: Once the console is running and connected to the WYSIWYG drawing, operate the console as you would in the live venue. Run up channels and record cues using the console. Remember that WYSIWYG does not store or save cues—this is done in your console’s memory. Once the console information is set up, you can set WYSIWYG to automatically connect to the console every time the file is opened.

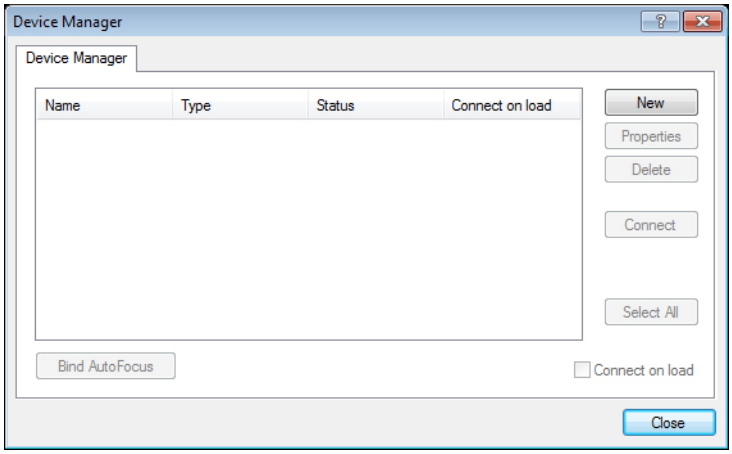
Note: You can also perform the above procedure through the Device Manager toolbar. For details, see “Device Manager toolbar” on page 529.

To connect the Image Console

The Image Console is produced by CAST Software and ships with WYSIWYG. It acts like a lighting console, media, or video server, enabling you to control color, intensity, and the image display of all patched LED surfaces that you have created with the LED Wizard.

1 From the **Live** menu, choose **Device Manager**.

Result: The *Device Manager* dialog box is displayed.



2 Click *New*.

Result: The console library is displayed.

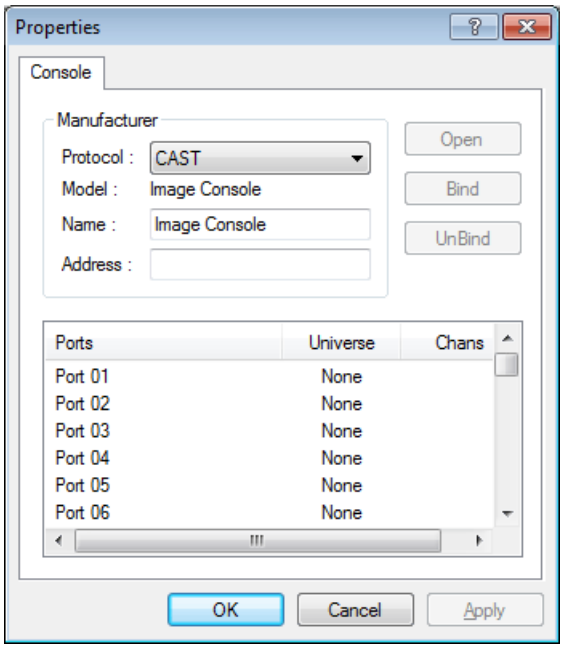
3 Double-click **Consoles** > **Manufacturer** > **CAST Software**, and then highlight **Image Console**.

4 Click *Insert*.

Result: The console appears in the *Device Manager*.

5 With the console name still selected, click *Properties*.

Result: The *Properties* dialog box is displayed.



6 Leave the defaults for *Protocol*, *Model*, and *Name*.

7 In the *Address* box, type the designated address of the console. Note that this address must match the *Device Address* that you type in the Image Console window. For details on configuring the Image Console, see “To configure the Image Console” on page 535.

8 A list of the output ports from the console appears in the *Ports* list. You must bind these outputs to WYSIWYG patch universes. Double-click the appropriate port to open the Patch Universe Select window.

9 Highlight the appropriate patch universe, and then click *Select*.

10 Repeat the above steps to bind all required ports for your LED grid to their appropriate patch universes.

Note: You cannot control a channel count exceeding that of the WYSIWYG Perform package that you purchased - 500, 1000, 2000, 5000, and so on.

11 Click *OK* to close the *Properties* box.

12 Click *Connect*.

Result: The status of the console changes to “connected.”

13 Click *OK* to exit the Device Manager. You can now control the patched LED grid with the connected Image Console. For details, see “To control a patched grid of LEDs with the Image Console” on page 538.

Device Manager toolbar



The Device Manager toolbar simplifies your regular device connect/disconnect operations. You can use the drop-down on this toolbar to select a device (Consoles, Devices, Networks), and then click the green connect icon or the red disconnect icon.

The “All Devices” option in the drop-down enables you to easily connect/disconnect all devices with one click without opening the Device Manager.

Working with consoles

Introduction

This section contains instructions for editing the console settings, as well as using consoles to control various types of elements, such as video and motion axes.

Note: For information about the mapping of images onto patched light-emitting surfaces, see “Image Console” on page 535.

To set the console to connect on load

1 From the **Live** menu, choose **Device Manager**.

Result: The *Device Manager* dialog box is displayed.

2 Select the console.

3 Select the *Connect on load* check box.

4 Click *OK* to close the *Device Manager*.

To edit the patch binding settings

1 From the **Live** menu, choose **Device Manager**.

Result: The *Device Manager* dialog box is displayed.

2 Select the console.

3 Click *Properties*.

4 Select the port for which you want to change the patch binding.

5 Click *Unbind*.

6 Rebind as desired (follow steps 9 through 12 in “To connect to a console” on page 526).

To remove a console from the Device Manager

1 From the **Live** menu, choose **Device Manager**.

Result: The *Device Manager* dialog box is displayed.

2 Select the console that you want to remove.

3 Click *Delete*.

Result: The console is removed and all patch binding information is lost.

Controlling a DMX patched video source with a console

Note: If the video control is patched and DMX connected in Live Mode, then you will not be able to control the video using any of the Video Designer Tool commands in Design mode. Video sources can only be controlled by a designer tool when the DMX source is disconnected; therefore, you must first disconnect the applicable console device from Live Mode before using any of the Video Designer Tool commands.

After you have patched the control of a video source in Data mode, you can use the applicable console device to control the progress of the video in Live mode.

In WYSIWYG, you patch the *control* of a video source, not the actual video. This means that you can use the console device to control the progress of a live video stream or a video file by making the video play, pause, or rewind; you cannot use WYSIWYG to change different aspects of the video itself.

To control the video source, the DMX patch is allocated three channels with the following controls:

- **First channel (Intensity)**: The video is off if the value is set between 1—128 (below 50%), it is on if the value is between 128—255 (above 50%).
- **Second channel (Control)**: The video is controlled by the position that you set on the third channel if the value is set between 1—84 (between 1% and 33%), it is paused if the value is set between 85—169 (between 33% and 66%), and it plays if the value is set between 170—255 (between 66% and 100%).
- **Third channel (Position)**: If you have set a value within the bottom third level of the second channel (between 1% and 33%), then you can use the third channel like a slider, advancing the video either backward or forward to the exact position that you want to see. If the value on the second channel is set to any level above 84 (between 33% and 100%), then the third channel is disabled.

To control a DMX patched video source with a console

1 Ensure that the console device is connected to the patched video source. For details on connecting a console, see “To connect to a console” on page 526.

- To play the video from start to finish, perform the following steps:

a. Set the first channel to a value between 128—255 (above 50%).

b. Set the second channel to a value between 255—170 (between 100% and 66%).

- To pause the video, perform the following steps:

a. Set the first channel to a value between 128—255 (above 50%).

b. Set the second channel to a value between 255—170 (between 100% and 66%) and play the video to the spot where you want to pause it. Then move the second channel to a value between 169—85 (between 66% and 33%) to pause the video at this spot.

- To control the progress of the video manually, perform the following steps:

a. Set the first channel to a value between 128—255 (above 50%).

b. Set the second channel to a value between 84—1 (between 33% and 1%).

c. Slide the third channel up and down to control the progress of the video, forward and backward.

Controlling a DMX patched motion axis with a console

After you have patched a motion axis in Data mode, you can use the applicable console device in Live mode to control the movement of any objects that are attached to the axis.

To control the object, the DMX patch is allocated two channels with the following controls:

- **First channel (Move)**: The first channel is for coarse movement and can be used to advance the object quickly to any position along the entire path, from start to finish.
- **Second channel (Move Fine)**: The second channel is used for further refining the object’s position that you set with the first channel. Adjust the slider on the second channel to move the object very slightly either forward or back from its

current position. The movement achieved with this channel is so fine, that it is best viewed on linear axes that are quite long. Note that for rotation axes, the movement is so slight, that you may not be able to see it.

To control a DMX patched motion axis with a console

1 Ensure that the console device is connected to the patched motion axis. For details on connecting a console, see “To connect to a console” on page 526.

2 To move the object quickly along the motion axis, adjust the slider on the first channel, stopping at the position where you want to leave the object.

3 To move the object in fine increments either forward or back from its current position, adjust the slider on the second channel.

Tip: To view fine movement, the object must be attached to a very long linear axis, or be set to rotate in a very wide arc. It is also a good idea to zoom in quite close to the object before moving it.

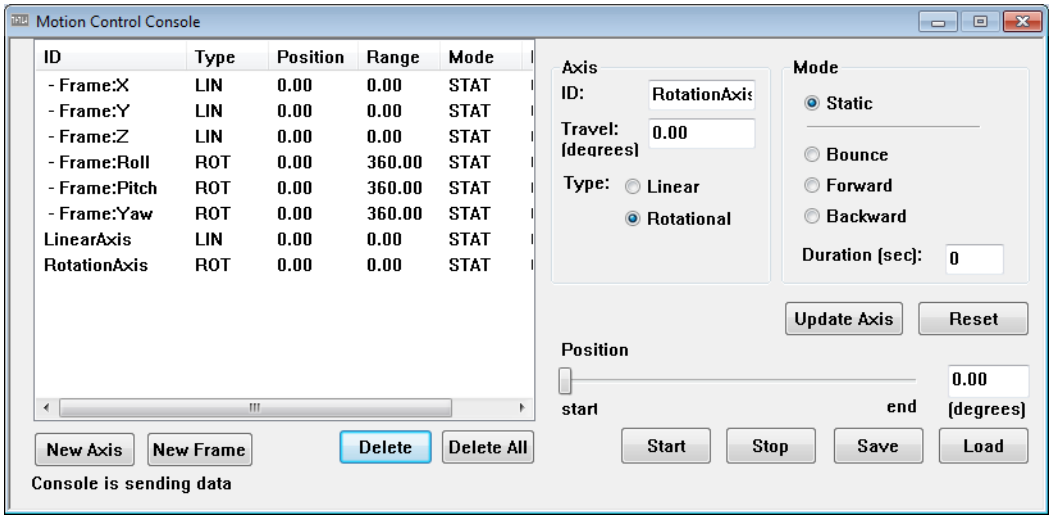
Controlling a motion-control patched motion axis with a motion control system

After you have patched a motion axis in Data mode, you can use the applicable motion control system in Live mode to control the movement of any objects that are attached to the axis.

Based on the type of system that you are using, you can control the object in different ways. The following procedure outlines how to control the object(s) with WYSIWYG’s Motion Control Console.

To control a motion-control patched motion axis with a motion control system

1 Launch the Motion Control Console.



2 Ensure that the Motion Control Console (or the motion control system if you are using one) is connected to the patched motion axis. You connect the Motion Control Console (and motion control systems) in much the same way as you connect consoles. For details, see “To connect to a console” on page 526.

3 To change the settings of a patched motion axis, highlight the axis in the left pane, and then choose from the following settings:

- **ID box**: To change the ID of the motion axis, type the new motion control ID.

- **Travel box**: Type the distance (in meters for linear axes, in degrees for rotational axes) that you want the object to travel along or around the axis. This can be the full length/angle of the axis, or only a portion of it if you do not want the object to travel along/around the full path. For linear axes, if you type a value that is longer than the actual axis, the object stops at the end of the physical axis.
- **Type**: If you change the axis type so that it does not match the axis that you have drawn (for example, if the actual axis is linear, but you choose rotational here), the cell in patch view turns to red.
- In the Mode area, select the type of movement for the object:
  - **Static** Select this option if you want to specify the precise location of a static object on the motion axis. You can then use the position slider at the bottom of the window to adjust the position and control the movement of the object manually.
  - **Bounce** Select this option if you want the object to move forward and backward in a continuous loop along the motion axis.
  - **Forward** Select this option if you want the object to move forward along the axis from start to finish, and then start over again at the beginning in a continuous forward loop.
  - **Backward** Select this option if you want the object to move backwards along the axis from finish to start, and then begin over again in a continuous backwards loop.
- **Duration box**: If you have chosen any *moving* Mode value (that is, any value *except* Static), you can specify the length of time (in seconds) over which you want the full range of motion to take place. The larger the number you type in this box, the slower the object moves.
- **Position slider**: For all moving modes (bounce, forward, backward), the slider indicates the position of the object when the console is sending data to WYSIWYG. For the static mode, while the console is started, drag the slider to to adjust the position of the object on its axis. The position value changes in the box in the left pane.
- **Position box**: While the console is sending data to WYSIWYG, for all moving modes, this box shows the progress of the object's movement along its axis; for the axes in the static mode, you can type the precise location of the object on the axis.

4 Click *Update Axis* to view your new settings.

5 Click *Save* to save the changes.

Note: If you make any changes to the axis settings and save the changes, you can revert to the previous settings by clicking *Load*.

Connecting with a Wyg-it 2

WYG-it 2 is CAST Software's DMX interface device. You require a DMX interface to connect a DMX console to WYSIWYG. Each WYG-it 2 has 2 DMX inputs that enable you to visualize up to 1024 channels of DMX. The WYG-it 2 also has 2 DMX through ports that allow you to connect the console both to WYSIWYG and to the physical lighting equipment at the same time. WYG-it 2 is equipped with the necessary outputs to send Autofocus commands.

You can connect the WYG-it 2 to your computer by inserting the WYG-it 2 USB cable into an available USB port. Once connected to your computer, the WYG-it 2 does not need an external power supply.

Note: When WYG-it 2 is installed, the WYG-it DMX Interface configuration panel is accessible in the Control Panel of your computer. The WYG-it 2 name recorded in this configuration panel is the value that you must type in the *Address* box of the Device Manager Properties window.

Mini Console

WYSIWYG contains an application that simulates a small console, called the Mini Console. You can use this application in lieu of a real console. To use the Mini Console, run the Mini Console application, found in the WYSIWYG section of the Windows Start menu. Connect as if you are using an external console.

The Mini Console gives you control of the first 25 channels on a single DMX universe but does not record or playback cues.

Image Console

Introduction

The Image Console is a control device much like the existing Mini Console, only specifically for controlling LED elements.

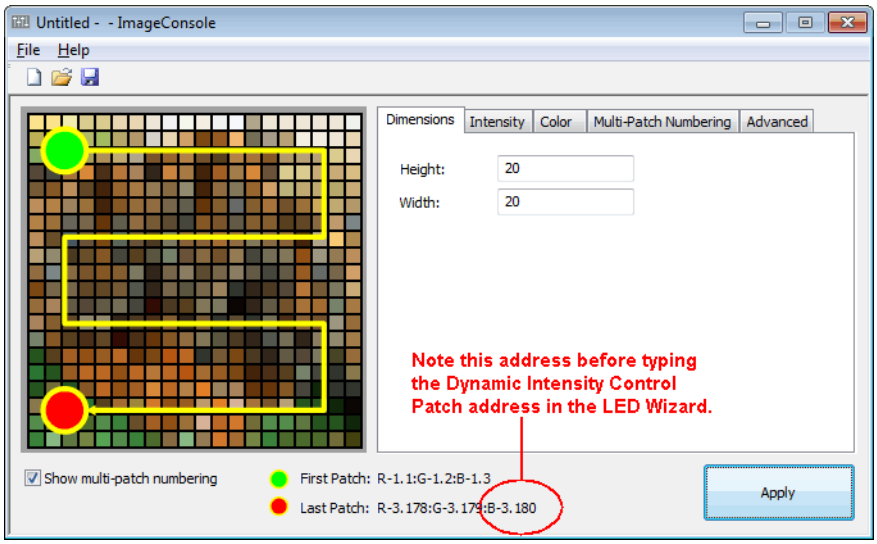
It acts like a lighting console, media, or video server, enabling you to control color, intensity, and the image display of all patched LED surfaces that you have created with the LED Wizard.

This console enables you to control and test color, intensity, and image mapping independently or simultaneously in Live mode.

Note: It is recommended that you configure the Image Console first and then use the LED Wizard to create the LED grid that will display the image.

If you are going to choose multi-patch for color, single patch for intensity *and* you are going to use the Image Console to display an image on the LED grid, then it is recommended that you note the last patch address for intensity that is displayed in the configured Image Console and type this same address for the Dynamic Intensity Control Patch in the LED Wizard.

In the following example of a configured Image Console, the last address for intensity is 193, so this is the value you would type in the LED Wizard for the Dynamic Intensity Control Patch. For details on using the LED Wizard, see “LED Wizard” on page 322.



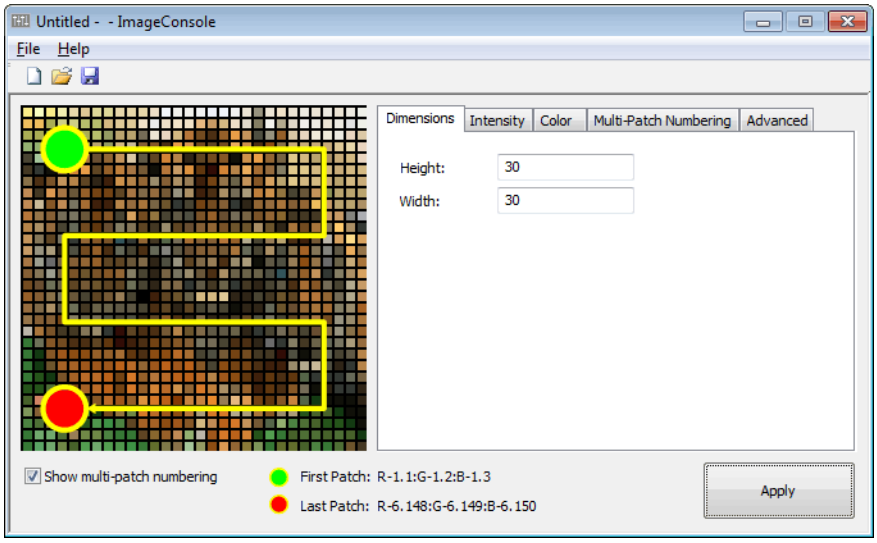
To configure the Image Console

Note: After you configure the Image Console, you configure and patch a grid of light-emitting surfaces using the LED Wizard. For details, see “To use the LED Wizard” on page 322.

For optimum image viewing, the patch settings that you have defined for both intensity and color in the Image Console must match those that you define for the LED grid. For example, if you choose multi patch for both intensity and color in the Image Console, then you must also choose multi patch for both properties in the LED Wizard.

1 From the **Start** menu, choose **WYSIWYG** > **Image Console**.

Result: The Image Console appears.



2 Click *Show multi-patch numbering* if you have enabled multi-patch and want to see a graphical representation of the numbering of the surfaces in the grid, either along columns/rows or in a serpentine pattern.

Note: The graphic appears only after you have chosen an image and clicked *Apply*.

3 Complete the following fields on the *Dimensions* tab:

- *Height*: Type the number of rows that you are going to configure in the LED grid.
- *Width*: Type the number of columns that you are going to configure in the LED grid.

4 Click the *Intensity* tab and complete the following fields:

- *Enable Dynamic Intensity*: Click this check box to control the intensity of the light emitting surface in Live Mode when connected to a console. Then choose the patch properties below.
- *Intensity slider*: Use the slider to select the maximum intensity value of the light-emitting surface when connected to a console and viewed in Live mode, with 255 representing the brightest possible intensity. This value enables you to control the scale of the intensity over a series of light-emitting surfaces. (When you disconnect from the console, the intensity is fixed at the static value.) Note that the DMX intensity value is treated as a percentage of the dynamic intensity value. For example, a DMX value of 127 results in 50% of the maximum intensity value that you set.
- *Single Patch*: Select this value to apply a single address that controls the intensity of the surface as a whole, and then choose whether you want to use the Default Single-Patch Address or specify the port and channel of the single patch address.
- *Multi Patch*: Select this value if you want to use different addresses for each individual element in the grid. With multiple addresses, you can make different parts of the grid have varying intensity levels.

5 Click the *Color* tab and complete the following fields:

- *Enable Dynamic Color*: Click this check box to control the color of the light-emitting surface in Live Mode when connected to a console. Then choose the patch properties below.

- *Single Patch*: Select this value to apply a single address that controls the grid as a whole, and then choose the color properties. For example, if you choose red from the drop-down color-picker, every surface in the entire grid will be red. You could use this setting to create a string of holiday lights with the same color, and then control the intensity of the lights as one string to turn them all on or off simultaneously. Then choose the default single patch address or specify an address.
- *Multi Patch*: Select this value if you have chosen different addresses for each individual element in the grid. When you choose multi patch, you can apply a graphic to the grid, with the colors in the graphic mapped to each of the grid's surfaces.
  - *Image*: Click the ellipsis button (...) to browse for the image that you want to map to the LED grid in your plot. You can choose from multiple graphic formats such as .gif, .jpg., or .bmp. For optimum results, the image size should be proportional to the grid size. For example, if your grid is 20 columns wide by 20 rows high, then your image should be perfectly square. If your grid is 30 columns wide by 20 rows high, then your image size should be in the ratio of 3 X 2 (for example 600 X 400 pixels).
  - *Mode*: Choose how the image is mapped to the surfaces in the grid. The *1:1* option is ideal for images that have the exact same dimensions as the grid on a pixel per row/column mapping (for example, if your grid is 20 x 20, then the image should also be 20 x 20 pixels). Choose *Stretch* to have the image stretched over the surfaces of the grid. Note that based on the image size, there might be distortion. Choose *Stretch Preserving Aspect Ratio* to stretch the image over the grid and keep its original aspect ratio. Note that this option is only suitable when the grid shape and image shape are the same. For example, if the image is rectangular, but the grid is square, some of the surfaces in the grid will remain blank because the image will not stretch to cover them.

6 Click the *Multi-Patch Numbering* tab and complete the following fields:

- *Start From*: For multi-patch, choose where you want to start the numbering of the surfaces in your grid.
- *Number Along*: Select the direction of the numbering, either along the rows or columns of the grid.
- *Alternate direction for each row/column*: Click this check box to have the numbering progress in a serpentine pattern throughout the grid.

Note: Click *Apply* to see a preview of your numbering choice.

7 Click the *Advanced* tab and complete the following fields:

- *Device Address*: Type the address of the Image Console. This name must match the Image Console address configured in the Properties window of the Device Manager. For details on using the Device Manager to connect the Image Console, see "To connect the Image Console" on page 528.
- *DMX Channels per Port*: Type the number of DMX channels required per port. Leave the default value of 512 for DMX universes. If you are using a universe with a greater number of ports (for example, EDMX), type the number of channels in this box. If you are using a protocol that supports more than 512 channels per port, type the value here.

8 Click *Apply*.

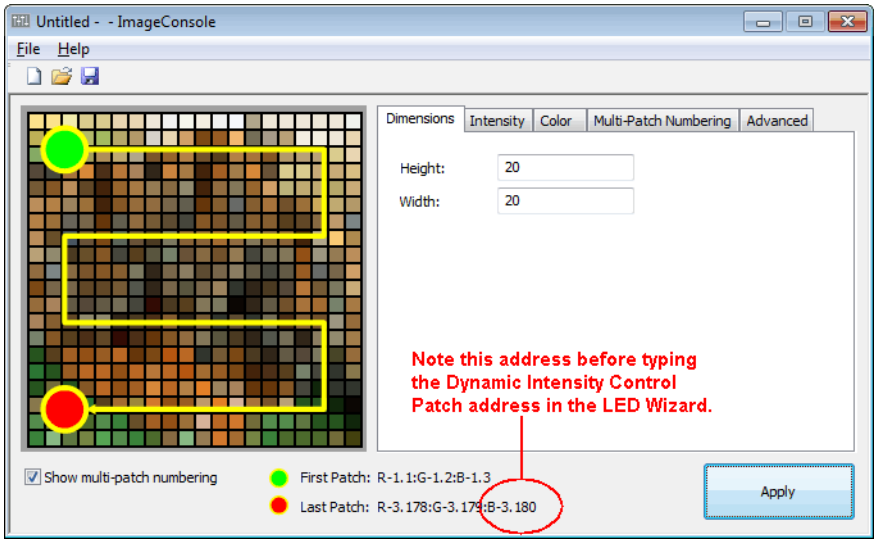
Result: A preview of the grid appears in the left pane. To change the look of the grid, adjust any of the values, and then click *Apply* again.

9 To save your settings so you can open the Image Console and use the same properties at any time, click the Save icon, and then type a name for the console.

What's next?

If you are going to choose multi-patch for color, single patch for intensity *and* you are going to use the Image Console to display an image on the LED grid, then it is recommended that you note the last patch address for intensity that is displayed in the configured Image Console and type this same address for the dynamic Intensity Control Patch in the LED Wizard.

In the following example of a configured Image Console, the last address for intensity is 180, so this is the value you would type in the LED Wizard for the Dynamic Intensity Control Patch. For details on using the LED Wizard, see "LED Wizard" on page 322.



To control a patched grid of LEDs with the Image Console

There are three prerequisites to this procedure:

- First you must configure the Image Console.
- Second you must create and patch an LED grid. For details, see "To use the LED Wizard" on page 322.
- Lastly you must connect the Image Console. For details, see "To connect the Image Console" on page 528.

1. Click Live mode > Shaded tab and adjust the view so you can clearly see the grid that you have created.
2. From the **Start** menu, choose **WYSIWYG** > **Image Console**.
3. To open a configured and saved console, click **File** > **Open**, and then navigate to the .lxi file containing your saved console properties. If you have not yet configured your Image Console with the image you would like to display, perform the steps in the procedure "To configure the Image Console" on page 535.
4. When you are satisfied with your settings, click *Apply*. Note the appearance of the LED grid in the Shaded tab. Adjust the properties of the Image Console as desired, and then click *Apply* again.

AutoPatch

Introduction

AutoPatch is a protocol by which WYSIWYG can transfer the patch hookup to a compatible console. AutoPatch is only available in WYSIWYG Perform.

You must be connected to a compatible console for this feature to be enabled. All ports for which you want the patch information to be transferred must be properly bound. Only the hookup information in patch universes bound to the console's outputs will transfer.

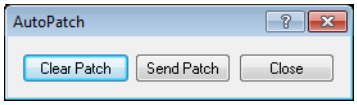
The patch information is transferred over Ethernet.

To perform an AutoPatch

1 Connect to the console as described in "To connect to a console" on page 526.

2 From the **Live** menu, choose **AutoPatch**.

Result: The *AutoPatch* dialog opens.



3 Click *Send Patch*.

Result: WYSIWYG assembles the patch information for all the fixtures in the plot and then attempts to send the relevant patch information to each connected AutoPatch-capable device.

Notes:

- A fixture's patch information is transmitted to a console only if the following conditions are met:
  - the fixture has at least one part that is patched to a patch universe
  - the patch universe is bound to the console
  - the console is connected to WYSIWYG through the *Device Manager*
- Some fixtures might have parts patched to two (or more) different consoles. In this situation, both consoles will receive the patch information for that fixture, but the port numbers given for each part will be set appropriately for each console.
- It is recommended that you click *Clear Patch* prior to sending the patch to clear out the existing patch.

4 Click *Close*.

To clear the patch

1 Connect to the console as described in "To connect to a console" on page 526.

2 From the **Live** menu, choose **AutoPatch**.

Result: The *AutoPatch* dialog opens.

3 Click *Clear Patch*.

Result: WYSIWYG prompts each AutoPatch-capable device to clear the patch for each port that is bound to a WYSIWYG patch universe.

4 Click *Close*.

Visualization

Introduction

There is a limitation on the number of DMX channels that can be simulated in Live mode. This limitation is based on the level of WYSIWYG Perform that you have installed on your computer.

WYSIWYG Perform is available in 500, 1000, 2000, 5000 and 50000 channel variants. With each of these variants, the following channels of simulation are available:

- WYSIWYG Perform 500=512 channels
- WYSIWYG Perform 1000=1024 channels
- WYSIWYG Perform 2000=2048 channels
- WYSIWYG Perform 5000=5120 channels
- WYSIWYG Perform 50000=51,200 channels

In Live mode, WYSIWYG will not simulate the DMX values being received on channels that exceed the Perform channel count of your system. The unused channels display on the status bar tracks the remaining channels for simulation, based on the level of your Perform system.

Note: Fixtures on layers that are not visible or not included in the current scene are not counted.



For more information on the status bar, refer to “Status bar” on page 68.

Simulation options

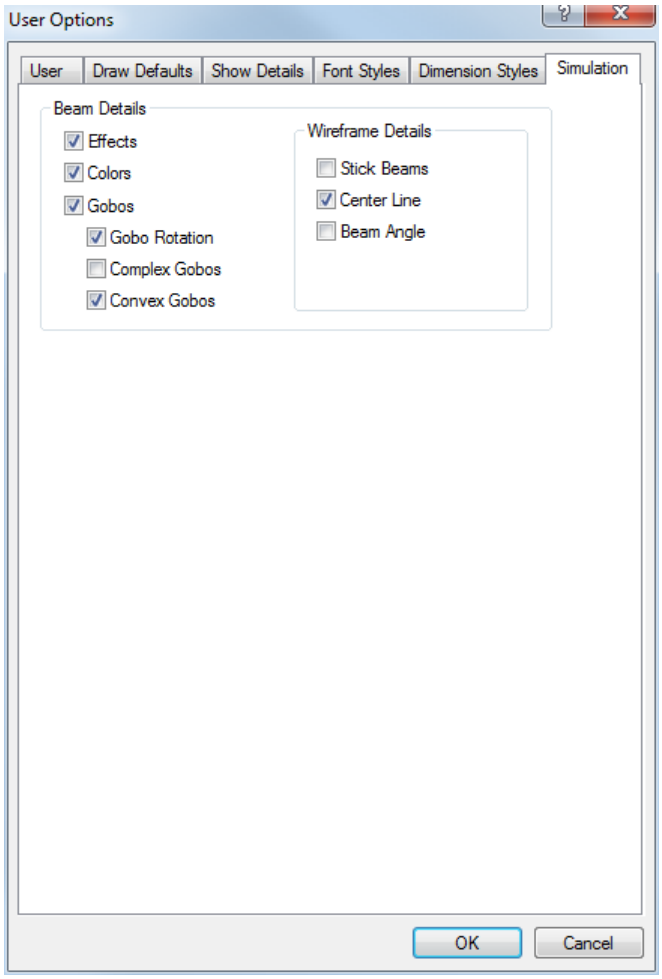
The Simulation options affect how fixtures respond in Live mode and how the beams are displayed in shaded views. Turning options off will yield faster simulation performance while compromising aesthetics. For optimal results, increase the processing power of your computer.

To change the simulation options

1 From the **Options** menu, choose **User Options**.

Result: The *User Options* dialog box is displayed.

2 Click the *Simulation* tab.



3 Make your selections, and then click *OK* to exit.

- *Effects:* To simulate effects such as prisms, select the *Effects* check box. This might increase the number of lines that WYSIWYG has to draw, thus slowing down processing power.
- *Colors:* Deselect the *Colors* check box to ignore color settings and have all beams displayed as open-white.
- *Gobos:* Select the *Gobos* check box to simulate gobo details. Gobo wheels often take a lot of processing power and can slow down the refresh rate of other fixtures' updates unnecessarily. If you are concerned about cue timing, set the scene first with gobos, and then turn off this option to obtain a faster refresh rate.
  - *Gobo Rotation*: To simulate gobo rotation, select the *Gobo Rotation* check box. This option can slow down the refresh rate substantially as gobos can be rotating even when the fixture's intensity is at zero.
  - *Complex Gobos:* Select the *Complex Gobos* check box to simulate complex gobo fixtures. Most gobos are drawn in a complex and a simplified form. The complex forms are far too slow for real time simulation and should only be used in rendering.
  - *Convex Gobos:* Select the *Convex Gobos* check box to simulate convex gobo fixtures. Some complex gobos that include shapes similar to a "C" have to be pre-processed to make them appear correctly. This can often reduce the refresh rate.

- *StickBeams*: Select the *StickBeams* check box to draw the center of the light beam for each fixture. This is the quickest way to simulate moving lights with the least beam clutter.
- *Center Lines:* Select the *Center* check box to draw a point from the fixture to the point on the stage where the center of the beam hits. If the fixture has CMY color mixing, the line will change color to match the output from the console.
- *Beam Angles:* Select the *Beam Angles* check box to display the photometrics of the fixture's beam angle instead of the field angle. The angle that is displayed is the measurement in degrees from the center of the beam to where the light level drops off to 50% of the peak. For more information on photometrics, refer to "Photometrics tab" on page 299.

Note: Also noteworthy are the properties of shaded views. These are discussed in "Shaded views" on page 78.

Autofocus

Introduction

Autofocus is the protocol that WYSIWYG uses to “talk back” to a compatible console. Using the mouse in a Wireframe view or using the concept shortcuts and design tools at any time, you can direct the console to select fixtures, set focus, and control intensity, iris, and color. Autofocus is connected through MIDI, Serial, or Ethernet.

The following consoles are compatible with the Autofocus protocol:

- Flying Pig Systems WholeHog II
- High End Systems WholeHog III
- Compulite Vector
- ETC Congo, Pronto, Prisma NT, Focus NT
- MA Lighting grandMA (all)
- LeHigh Rendition and Millenium
- Rosco Horizon software
- Chamsys MagicQ
- LightFactory software
- SGM Regia (all)

For detailed instructions on setting up AutoFocus to work with the HogII, refer to “Setting up AutoFocus to work with the HogPC” on page 571.

Assigning Spot Numbers to fixtures

In addition to assigning DMX addresses to fixtures, you must assign specific Spot Numbers to them in Data mode. The Spot Number that is assigned to the fixture in WYSIWYG must match the Fixture ID/Spot Number assigned to the fixture in the console.

Note: Unless you assign a Spot Number, you cannot control the fixture using AutoFocus.

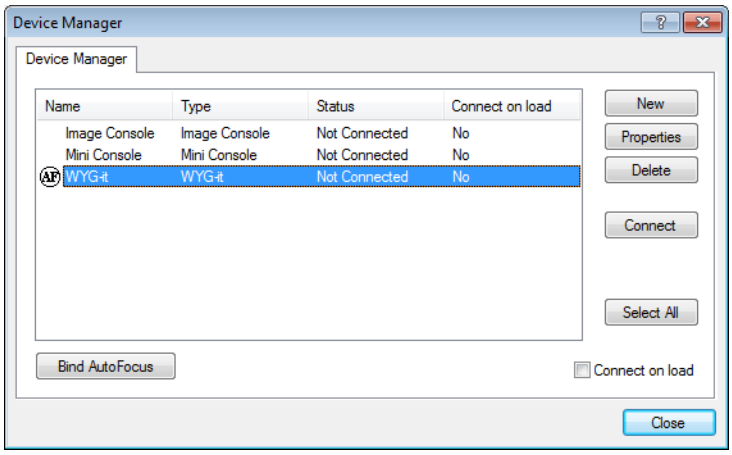
To bind Autofocus to the console

1 Follow steps 1 through 14 of “To connect to a console” on page 526.

2 Click *Bind Autofocus*.

Note: The *Bind Autofocus* button is only available when a compatible console is selected in the Device Manager.

Result: The autofocus icon appears beside the name of the console in the Device Manager.



3 Click *Connect*.

4 Click *OK* to close the Device Manager.

Autofocus tools and menu commands

To select fixtures

The procedure for selecting fixtures in Live mode is the same as selecting fixtures in Design mode using the design tools and Concepts shortcuts. For more information, refer to “Using the Design tools” on page 475.

To use the Autofocus tools

The procedure for using the autofocus tools is the same as using the design tools. For more information, refer to “Using the Design tools” on page 475.

To use the Autofocus menu commands

Right-click on a selected fixture or fixtures to open a menu with the following Autofocus commands:

- **At Level and Focus**: Sets the intensity of the selected lights on the connected Autofocus-compatible console to the level last specified by the Intensity command, and then launches the Focus command.
- **Focus**: When you click in a Wireframe view, the selected automated fixtures will focus (if they can) to that point. Click and drag to position them in real-time. Change the missing focus coordinate to alter the height at which you want to focus the fixtures.
- **Intensity**: Reveals a sub-menu where you can specify the level to set the intensity for the selected fixtures. The first option in the list is the last level you specified and by default is set to **Full**. You can also set the level to half and out.
- **Iris**: Sets the iris to either tight or wide. There is also a Specify option where you can set the iris to any percentage. The Specify value is retained for the session.

- **Color**: Fixtures that can mix color can be directed to either a palette color or an approximation of a manufacturer's gel color. You can specify and save palette colors using the custom color creator or Palette shortcuts.

Tips:

- You can repeat Autofocus commands by touching the Space bar. This way, you can select one fixture, adjust it, select another, and then press the Space bar to perform the same operation repeatedly.
- Alter the missing focus coordinate to focus fixtures at different heights.
- When focusing fixtures, you can click and drag in the wireframe view. If you are using WYSIWYG in the theatre, launch this command, and then look to the stage to position your lights.
- Working in an Isometric view, you can use the snap tools to focus fixtures to points on your set pieces.
- If you set the intensity to a specific value, the right-click menu option **At Full** and **Focus** changes to At ##% and Focus where ## is the specific value that you set.
- Select multiple fixtures by dragging a window around them. If you drag the window from right to left across projected beams, you can select the fixtures that produced the beams.
- To select fixtures by type, draw a box using the right mouse button. You will be prompted to isolate one or more types of fixtures.
- Use the Concept shortcuts to create and select groups of fixtures.
- Use the Palette shortcuts to save and send color commands.

Rendering

Introduction

At any time in Live mode you can generate a rendering of a lighting cue or static look. The Render Wizard calculates the rendering based on the DMX values being received at the moment when you click *Finish*.

To render a lighting look

1 In the working area of the screen, display the lighting look that you want to render.

2 From the **Design** menu, choose **Render**.

Tip: You can also click the *Render Wizard* tool in the *Design* toolbar. 

Result: The Render Wizard opens and guides you through a series of options. Click *Next* and *Back* to navigate through the Render Wizard.

3 Click *Finish*.

Result: WYSIWYG generates the rendering of your lighting look or cue. If the rendering is saved internally, it is available as an image in Presentation mode.

Note: For more information, see “Rendering” on page 497.

DMX Camera

Introduction

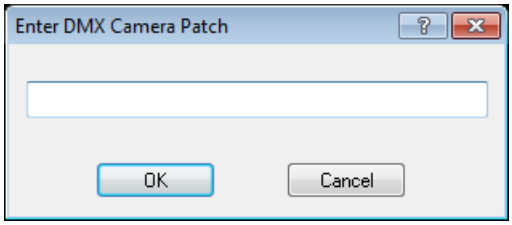
With the DMX Camera utility, the camera's orientation (position and target), zooming, and roll features can all be manipulated and recorded to create a realistic rendering of a professional, fully edited film clip.

It is important to distinguish between DMX Camera and cameras drawn in the CAD mode of WYSIWYG. Drawn cameras provide a single viewpoint; the camera can be moved, but only by one parameter at a time and there is no possibility to track or record camera movements. DMX Camera is not drawn in your plot and is completely maneuverable; each attribute is patched to a DMX channel and controlled through the console of your choice. Changes can be recorded on a separate cue list, if the console allows it; alternatively, camera changes can be recorded directly into previously created cues. The result is a recorded camera track that runs simultaneously with your show.

To use DMX Camera

1 From the **Live** drop-down menu, select **DMX Camera Patch**.

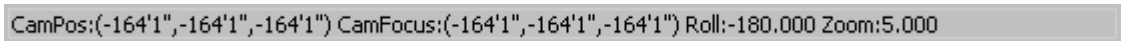
2 In the *Enter DMX Camera Patch* window, enter a new patch (for example, type Camera.1, where the "1" indicates the first channel number that the camera attributes will patch into).



3 In the Device Manager, bind the appropriate output of your selected console with the camera's patch universe, and then connect to the console.

4 In Live mode, click the *Shaded* tab.

Result: The position and focus point coordinates and roll and zoom angles of the camera display on the status bar.



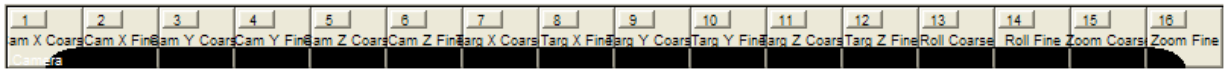
5 From the **Options** menu, choose **View Options**.

6 Select the *Options* tab.

7 From the *Camera* drop-down menu, select *DMX control*.

8 Click *OK*.

Result: You should now have direct control over the DMX camera. It may be helpful to record the camera patch as a pop-up frame in data to view the attributes that you are modifying.



To provide a fairly neutral starting point, it is recommended that the majority of the sixteen DMX Camera channels initially be set at a level of 50 per cent. The exceptions are "camera Y coarse", which should be set at 40 per cent, and "target Y coarse", which should be set at 60 per cent. Adjusting these two attributes will ensure that the camera and the target are not occupying the same point in three-dimensional space. "Personalities" may be created to reflect this.

The camera's initial position will be dictated by the document origin (not to be confused with the user origin, which can be altered at any time).

The 16 camera parameters are as follows:

1. camera X coarse
2. camera X fine
3. camera Y coarse
4. camera Y fine
5. camera Z coarse
6. camera Z fine
7. target X coarse
8. target X fine
9. target Y coarse
10. target Y fine
11. target Z coarse
12. target Z fine
13. roll coarse
14. roll fine
15. zoom coarse
16. zoom fine

Notes:

- The "coarse" parameter is defined as 100 metres/256 steps, or 0.39 metres per step.
- The "fine" parameter is defined as 100 metres/65536 steps, or 0.0015 metres per step.
- The zoom function of the DMX camera has a range of 5° to 100°.
- The "coarse" zoom parameter is defined as 95°/ 256 steps, or 0.37° per step.
- The "fine" zoom parameter is defined as 95°/ 65536 steps, or 0.00145° per step.

Patching camera paths

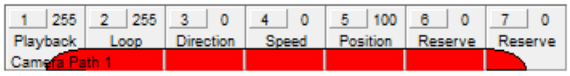
Introduction

Once you draw a camera path in CAD mode, you can patch it to a DMX universe to control the camera's movement along the path in Live mode.

For details on camera paths, see "Camera Paths" on page 493.

The Camera Path requires seven DMX channels, as shown in the DMX map below:

- Channel 1:Stop/Pause/Play... Stop: 0; Pause: 1-127; Play: 128-255.
- Channel 2:Looping... Enable: 0-127; Disable: 128-255.
- Channel 3:Direction... Forward: 0-127; Reverse: 128-255.
- Channel 4:Speed... x1: 0-127; x1/4: 128-159; x1/2: 160-191; x2: 192-223; x4: 224-255.
- Channel 5:Position (Scrubbing): controls the playback start position along the camera path. This is similar to dragging the marker in the timeline but relative to camera path's total spatial length; please note that it is not possible to 'scrub via DMX' when a Flying Camera is playing, only when it is paused (i.e., Channel 1 is receiving a DMX value of 1 through 127).
- Channel 6:Reserved.
- Channel 7:Reserved.



To patch a camera path

1. Select the Camera Path.
2. Right-click and select Properties (or press ALT+<enter>).
3. Click the Camera Path tab.
4. In the Patch area at the bottom, click the DMX Patch radio button.
5. Select an existing Universe and then enter a channel number or click the ellipsis button to create a new Universe.
6. Click *OK*.

Tips for controlling camera path playback via DMX

After binding the universe to which the camera path was patched to the connected console's output, set Channel 1's level to 128 or above to begin playback.

Pause playback by setting this channel's level to anywhere between 1 and 127. While paused, increase the level of Channel 5 to begin scrubbing along the Timeline.

Note: The flying camera will only move when scrubbed once the Position channel's level becomes higher than the position along the Camera Path where playback was paused. This means that if playback was paused around the (physical) middle of the camera path, Channel 5's level will have to reach above 127 (or so) before the Shaded View starts to update. Once the correct level is reached though, decreasing this channel's level will scrub the flying camera back towards the start point of its camera path.

Much like when scrubbing along the timeline by clicking and dragging the marker with the mouse, Direction is ignored. In other words, regardless of whether Channel 3 tells the flying camera to move forward or backward, increasing Channel 5's level will always move the flying camera towards the physical "end" point of its camera path, while decreasing this channel's level will always move the flying camera towards the physical "start" point of its camera path.

Using cues on a console, it is very easy to begin playback of a flying camera from a position other than the camera path's "start" point:

- Play; record Cue 1.
- Pause; record Cue 2 with a zero-count and make it auto-follow Cue 1.
- Scrub to the position where playback is to start; record Cue 3 with a zero-count and make it auto-follow Cue 2.
- Play; record Cue 4 with a zero-count and make it auto-follow Cue 3.

Using multiple cameras

When multiple camera paths are patched, starting playback for a camera path that's currently paused/stopped will "switch" to that flying camera immediately. To "switch back" to the flying camera that was playing initially, simply 'nudge' its Playback channel (i.e., set its level to something else within the "Play" range). This helps achieve 'cutting' between different camera shots.

To use multiple cameras

Using cues on a console, it is easy to Pause one camera, Play another, scrub the initial camera to a different position and then start its playback from that position:

1. Play the first camera; record Cue 1.
2. Play the second camera and Pause the first camera; record Cue 2. (Timing for Cue 2 depends on how far the first camera needs to move before cutting to the second.)
3. Scrub the first camera to a new position and 'nudge' the Playback channel for the second camera so it continues to play; record Cue 3 with a zero-count and make it auto-follow Cue 2.
4. Play the first camera; record Cue 4. (Timing for Cue 4 depends on where the second camera needs to get before cutting back to the first.)

Appendix A

Using the SuperPro Monitoring Tool



The Sentinel SuperPro Monitoring Tool is automatically installed with WYSIWYG. Using this tool, you can view and track license distribution information for the dongles on your network.

In this appendix

SuperPro Monitoring Tool 555



SuperPro Monitoring Tool

Introduction

As part of the WYSIWYG installation, a Sentinel SuperPro Monitoring Tool is included that enables you to view and track information about the dongles on your network.

With this tool you can:

- keep track of how many licenses are in use on your network.
- view the maximum number of licenses that can be obtained from each dongle and, therefore, the maximum number of users (both local and on the network) that can access WYSIWYG Learn or WYSIWYG Network.
- view the Sentinel Protection Servers that are currently installed and running on your network.
- view the addresses and version information for each Sentinel Protection Server.

The Monitoring tool is a convenient tool to use to keep track of license activity on your network. For example, you can use the Monitoring Tool to determine whether you have purchased enough licenses, based on license demand.

Note: The SuperPro Monitoring Tool is designed for use only with SuperPro-protected applications, such as WYSIWYG Learn or WYSIWYG Network, that are being run on a network.

To launch the Monitoring Tool

1. Navigate to the folder where you installed WYSIWYG (for example, C:\Program Files\WYSIWYG).
2. Open the Bin\Sentinel folder.
3. Double-click the *monitor* application.

   Result: The Monitoring Tool is launched, and you are prompted to set a protocol. The Monitoring Tool can monitor servers on the network that use any of the following protocols:

   - TCP/IP
   - NETBEUI
   - IPX/SPX
   - SAP
4. Select TCP/IP, and then click *OK*.

   Result: Servers using the selected protocol are displayed in the Monitoring Tool. WYSIWYG products run on TCP/IP networks only.

Changing the protocol

When you launch the SuperPro Monitoring Tool, you are prompted to set a protocol.

To modify the selected protocol

1 From the **File** menu, choose **Set Protocol**.

2 Select the protocol that you want to use.

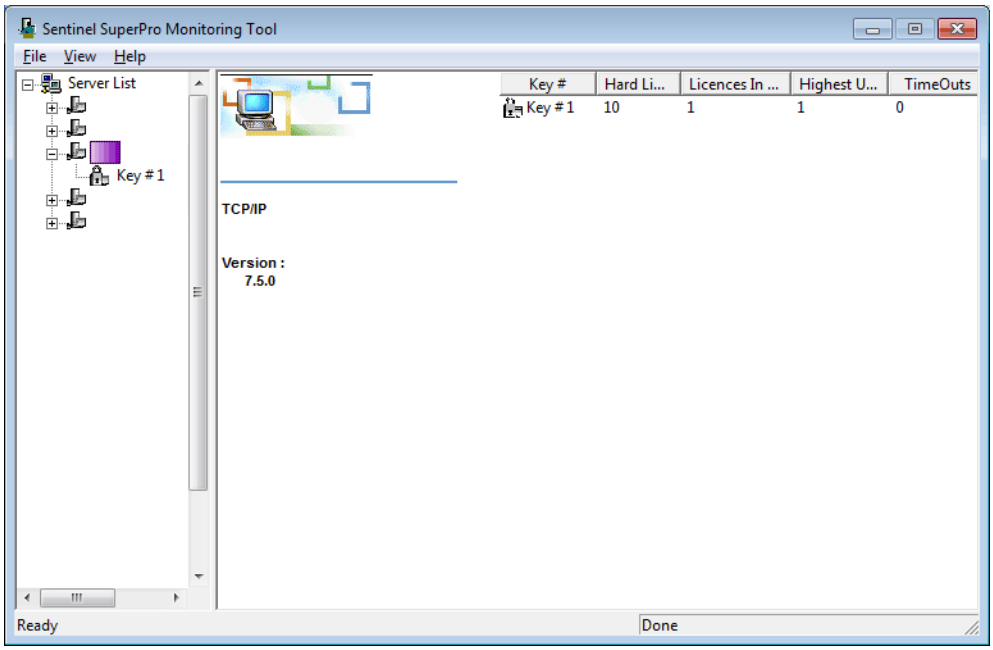
3 Click *OK*.

Result: Servers using the selected protocol are displayed in the Monitoring Tool.

Viewing servers on the network

When you launch the SuperPro Monitoring Tool, all Sentinel Protection Servers running on the network appear in the server list on the left. Click a server in the list to display the following information in the right pane:

- server name
- server's protocol
- server's address (for TCP/IP or IPX/SPX protocols)
- server's version
- key information for the dongle that is connected to the server



Key information

Information about each key (or dongle) connected to a server running on the network appears in the right pane when you select a server from the server list. Click on a key to obtain the following information:

- key number (a label used only for identification purposes)
- key's hard limit (that is, the maximum number of licenses that can be obtained from the dongle and, therefore, the maximum number of users that can access WYSIWYG Learn or WYSIWYG Network)
- number of licenses currently in use on the dongle
- highest number of licenses issued from this dongle
- number of time-outs recorded by the Server for Clients using licenses from the dongle

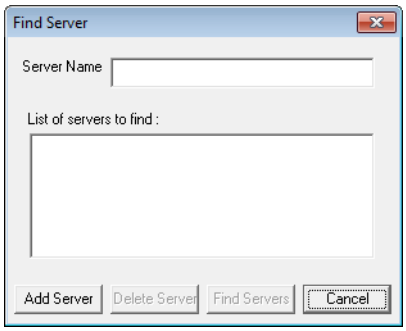
Searching for a server

If a server does not appear in the list, you can search for it on the network.

To search for a server

1 From the **View** menu, choose **Find**.

Result: The *Find Server* dialog box is displayed.



2 In the *Server Name* box, type the host name or IP address of the server that you are looking for.

3 Click *Add Server* to add the server to the list of servers to find.

4 Click *Find Server* to search for the server. If the server is found, it appears in the server list in the left pane.

Note: Once a server has been added to the list of servers to search for, it remains on the list until you remove it. To remove a server from the list, select the server, and then click *Delete Server*.

Refreshing the server list

You can either manually or automatically refresh the server list using the Monitoring Tool.

To manually refresh the server list

From the **View** menu, choose **Refresh**.

To automatically refresh the server list

1 From the **View** menu, choose **Refresh Rate.**

2 In the *Refresh After* box, type the refresh rate in minutes.

3 Click *OK*.

Result: The server list is automatically refreshed at the rate that you specify.

Additional help

For further assistance in using the Sentinel SuperPro Monitoring Tool, refer to the online Help available with the product.

Appendix B

Keyboard shortcuts and hot keys



Keyboard shortcuts and hot keys help you to access commonly used commands more quickly.

In this appendix

Keyboard shortcuts and hot keys 561

Keyboard shortcuts and hot keys

Keyboard shortcuts

There are many keyboard shortcuts that can save you time when using WYSIWYG.

Standardized WYSIWYG keyboard shortcuts are defined below.

Description	Keyboard shortcut
Help	F1
Displays context-sensitive help	Shift + F1
Repeat last command	F4 OR Spacebar
Refresh current view	F5
Relative or absolute coordinate toggle	F6
Ortho toggle	F8
Snap toggle	F9
Crosshairs toggle	F7
Interactive object creation toggle	F11
Save current document as...	F12
Repeat last command	Space
Select all	Ctrl+A
Select current layer	Ctrl+Shift+A
Create new layer	CTRL+Shift+L
Copy	Ctrl+C
Find	Ctrl+F
Group	Ctrl+G
Replace	Ctrl+H
Move	Ctrl+M
New document	Ctrl+N
Open document	Ctrl+O
Print current view	Ctrl+P
Rotate	Ctrl+R
Save current document	Ctrl+S
Ungroup	Ctrl+U

Description	Keyboard shortcut
Paste	Ctrl+V
Cut	Ctrl+X
Redo	Ctrl+Y
Undo	Ctrl+Z
Add to or remove from selection list	Ctrl+Click
Copy selected objects	Ctrl+Drag
Rotate selected objects	Alt+Drag
Mirror selected object	CTRL+W
Nudges selected objects by ten times the grid spacing	Alt+Arrow keys
Nudges selected objects by one grid space	Alt+Shift+Arrow keys
Displays properties of selected object	Alt+Enter
Move target	Arrow keys
Rotates around the drawing (in isometric view only)	Ctrl+arrow keys
Zoom in and out	Page Up/Page Down
Zoom fit	Z
Erases selected object	Del
Abort command	Esc
Finish current command	Enter

Hot keys

Hot keys are keyboard shortcuts that act as commands entered from menus or toolbars. They are categorized according to the view and timing of usage. For example, certain hot keys are only operational in plot views and only in the midst of performing an operation.

Default hot keys available in Plot Views

The following default hot keys are useful when working in CAD mode.

Start Operations

Description	Hot Key
Insert line	L
Insert riser	R
Insert cylinder	C
Insert pipe	P
Insert truss	T
Insert text label	B
Insert fixture	F
Insert point	.
Insert circle	O
Insert arc	U
Insert surface	S
Focus position	X
Insert screen	N
LED Wizard	W
Snap to grid	G
Snap to midpoints	M
Snap to endpoints	E
Snap to intersection	K
Zoom fit	Z
Modify view	V
ToolTips on/off	Y
Insert dimension	D

Mid Operations

Description	Hot Key
Abort mode	A
Finish mode	F
Close line	C
Quick increase	I
Quick decrease	D
Snap to grid	G
Snap to midpoints	M

Description	Hot Key
Snap to endpoints	E
Snap to intersection	K
Rotate truss counter-clockwise	Z
Rotate truss clockwise	X
Truss rotate from one end to the other end	S
ToolTips on/off	Y
Boolean subtract	CTRL+B
Boolean unite	CTRL+Shift+B
Distribute center	CTRL+D
Distribute between two points	CTRL+Shift+D
Focus selected fixtures	CTRL+F
Consolidate mesh	CTRL+Q
Trim line	CTRL+T
Extend line	CTRL+Shift+T
Extrude object	CTRL+E
Quick Tools	Q (fixtures are not selected)
Quick Focus	Q (fixtures are selected)

Default hot keys available in Simulation Views

The following default hot keys are helpful when working in Design and Live modes.

Start Operations

Description	Hot Key
At last and focus	L
Focus	F
At full intensity	1
At 50% intensity	5
Intensity off	0
Intensity specify	S
Iris wide	W
Iris tight	T
Iris specify	I
Zoom fit	Z
Modify view	V
ToolTips on/off	Y

Mid Operations

Description	Hot Key
Abort mode	A
Finish mode	F
ToolTips on/off	Y
Export image	E
Save view as New Camera	C
Camera lock	X

Default hot keys available in Layout View

The following default hot keys are helpful when working in Presentation mode - Layouts.

Start Operations

Description	Hot Key
Insert line	L
Insert rectangle	N
Insert round rectangle	U
Insert ellipse	E
Insert polygon	G
Insert text	T
Insert bitmap	B
Insert plot view	P
Insert data view	D
Insert report view	R
Insert spreadsheet	X
Open bound view	O
Bound view properties	S
Move to front	J
Move to back	K
Placeholders only	H
Print preview	W
Zoom fit	Z
Modify view	V

Mid Operations

Description	Hot Key
Abort mode	A
Finish mode	F

Default hot keys available in New Plots View

The following default hot keys are helpful when working in Presentation mode - New Plots.

Start Operations

Description	Hot Key
Snap to end-point	E
Snap to mid-point	M
Snap to grid	G
Snap to intersection	K
Insert solid line	L
Insect circle	O
Insect Arc	U
Insert text label	B
Insert symbol	S

Mid Operations

Description	Hot Key
Zoom fit	Z

Default hot keys available in Patch Views

The following default hot keys are helpful when working in Data mode - Patch.

Start Operations

Description	Hot Key
Console modify	C
Unpatch	U
Modify view	V
ToolTips on/off	Y

Mid Operations

Description	Hot Key
Abort mode	A
ToolTips on/off	Y

Appendix C

Using AutoFocus with the HogPC



AutoFocus is the protocol that WYSIWYG uses to “talk back” to a compatible console. One of the consoles that is compatible with this protocol is the HogII. The offline editor for the HogII, HogPC, is also compatible with the AutoFocus protocol, although it requires additional setup.

Normally AutoFocus is enabled through the WYG-it2 or other hardware interface. However, since the HogPC operates internally, you can establish a connection through software.

Note: HogPC and WYSIWYG must be running on the same computer to enable the connection.

In this appendix

Setting up AutoFocus to work with the HogPC 571

Setting up AutoFocus to work with the HogPC

Introduction

To establish a connection between AutoFocus in WYSIWYG and the HogPC, you must install and configure two additional pieces of software called MIDI YOKE and MIDI-OX. After this software has been configured, you can set up the HogPC and WYSIWYG to work with the new software.

Additional software

WYSIWYG can only send out MIDI data streams through the MIDI device set up in the Windows *Sound Devices* Control Panel. Similarly, HogPC can only listen to MIDI data streams that are fed into the computer through the MIDI device assigned by the same Control Panel. To establish a connection between WYSIWYG and HogPC, you must set up a third-party method.

One solution would be to use an external/physical device that can 'loop' the MIDI signal through it. That is, the device would take the signal coming from the computer on one port, and feed it back into the computer on another port.

A second solution would be to use a software driver that emulates the MIDI signal's 'looping' (the MIDI drivers that come with Windows do not support this feature.) This driver is called MIDI YOKE.

MIDI YOKE is a MIDI patch cable driver that enables you to establish a connection between applications that *output* MIDI data streams and those that *receive* MIDI data streams without the need for an external MIDI device. To obtain additional information on MIDI YOKE, refer to http://www.midiox.com/myoke.htm.

MIDI-OX is the application used to configure the MIDI YOKE port mappings/ assignments. It is also used to monitor MIDI data that passes through the MIDI YOKE driver. In the case of WYSIWYG, this is the MIDI data coming from WYSIWYG into the MIDI YOKE driver and out to the HogPC. To obtain additional information on MIDI-OX, refer to http://www.midiox.com.

Installing and configuring the additional software

MIDI YOKE

To install MIDI YOKE

1 Based on your operating system, download the MIDI Yoke installer from the appropriate link at the following Web site:

- Windows ME/XP: http://www.midiox.com/zip/myokent.exe

Follow the instructions at http://www.midiox.com/midiyoke_instxp.htm to install MIDI YOKE in a Windows XP Home environment. For all other operating systems, refer to the Installation section of the main MIDI YOKE Web page, http://www.midiox.com/myoke.htm.

Notes:

1. Do not change any of the settings in the configuration window that appears after installation is complete.
2. You must restart your computer to finalize the installation.

To configure MIDI YOKE

Once your computer has restarted, from the **Start** menu, choose **Settings**, and then choose **Control Panel**. Click the *Audio Devices* icon (or its equivalent name in your operating system), and then click the *Audio* tab. Select *MIDI Yoke NT: 1* as your MIDI Music Playback default device.

MIDI-OX

To install MIDI-OX

1 Download MIDI-OX by clicking http://www.midiox.com/zip/midioxse.exe.

2 To install MIDI-OX, double-click on the downloaded file and follow the instructions.

To configure MIDI-OX

1 Once the software has been installed, launch the application by double-clicking on the *MIDI-OX* icon located on your Desktop.

2 From the **Options** menu, select **Devices**.

3 Select *MIDI Yoke NT: 1* as your MIDI Input Device and *MIDI Yoke NT: 2* as your MIDI Output Device.

4 Click *OK*.

Setting up the HogPC

Note: For proper functionality, install the HogPC software on your computer *after* you install WYSIWYG. If the HogPC software was already installed before you installed WYSIWYG, repair the installation using the HogPC installer or the Add/Remove Programs utility in the Windows Control Panel.

To set up the HogPC

1 Launch HogPC.

2 Load the show for which you want to use AutoFocus to cue.

3 Right-click in the HogPC panel, and then choose *Config*....

4 Select *MIDI Yoke NT: 2* for Input, and *MIDI Yoke NT: 3* for Output, and then click *OK*.

5 In the HogPC panel, click *Setup* (next to the *Pig* button).

Result: The display changes in the first 'monitor' window.

6 Click *Input Panel*, and then click *MIDI In* at the top. (You can now close the Input Panel setup window, if desired.)

Setting up WYSIWYG

Due to some incompatibilities with certain motherboards, MIDI Drivers are not installed with WYSIWYG, even when you perform a full installation. Therefore, you must install them manually.

Note: Close WYSIWYG before starting the installation.

To install MIDI drivers

1. From the **Start** menu, choose **Settings**, and then choose **Control Panel**. Click the *Add/Remove Programs* icon.
2. Scroll down to WYSIWYG, and then click the *Change/Remove* button.
3. Once the installer launches, click the *Modify* option button, and then click *Next*.
4. From the *Select Components* window, select the *MIDI Drivers* check box.
5. Click *Next*.

   Result: Based on your method of installation, you are prompted for the installer file.

   a. If you installed WYSIWYG from a CD, you must place the WYSIWYG CD in your CD-ROM drive.

   b. If you downloaded the WYSIWYG installer from the Web, the install file must be present on your hard drive for the installation to complete.

      Please note that you will only be able to install the MIDI drivers from the installer that you originally used (for example, if you used the Web installer, you will not be able to install the drivers using the CD, and vice-versa).
6. Click *Finish* when the installation is complete.

To set up MIDI drivers

1. Launch WYSIWYG.
2. Load the file with which you want to use AutoFocus.
3. Switch to LIVE Mode, add the HogII in the Device Manager, and then bind the ports as you would normally.
4. Before closing the Device Manager, click *New*, and then select *Standard MIDI* from the *Devices* > *All* or from the *Devices* > *Manufacturer* > *Generic* tree.
5. Click *Insert*.

   Note: The Standard MIDI device appears in the list even if the drivers are not installed. Therefore, it is important that you follow the steps outlined above to ensure that the drivers are installed.
6. You now need to unbind AutoFocus from the HogPC, and then bind it to the Standard MIDI device. To do so, with *Standard MIDI* still selected, click the *Bind AutoFocus* button at the bottom of the Device Manager Window.

Testing the connection

Before performing this procedure, ensure that all three applications (MIDI-OX, HogPC, and WYSIWYG) are open.

To test the connection

1. In the Device Manager, select the Hog II, and then click *Connect*. The status changes to Connected.
2. Repeat step 1 for the standard MIDI device, and then close the Device Manager.
3. Resize your WYSIWYG window so that you can see at least part of the MIDI-OX window.
4. From the **Live** menu in Live mode, open all the Designer tools: Intensity, Focus, Color, Gobo, Prism, Zoom, and Iris.

5 In WYSIWYG, select at least one of the fixtures in your show, and then click *Full* in the Intensity tool.

Result: The selected fixture's intensity is displayed. In addition, HogPC's first monitor window should display *Intensity: Full* in the bottom left-hand corner.

Note: Even before you click *Full,* you should see various codes (MIDI data) displayed in MIDI-OX's Output Monitor if everything was set up according to the instructions above.

6 You can now minimize (not close) MIDI-OX, and start programming your show using WYSIWYG's AutoFocus features.

Appendix D

Connecting to an ArtNet Device



WYSIWYG contains drivers that enable you to “listen” to incoming DMX data when connected to the updated ArtNet network device in LIVE mode.

In this appendix

Connecting to an ArtNet Device 577

Connecting to an ArtNet Device

Introduction

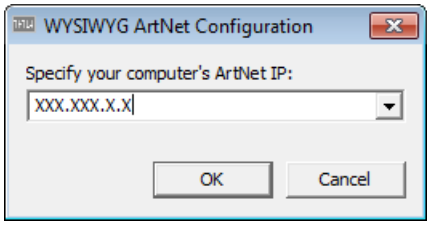
WYSIWYG contains drivers that enable you to “listen” to incoming DMX data when connected to the updated ArtNet network device in LIVE mode.

To connect to an ArtNet device

1 Click **Start** > **All Programs** > **CAST Software** > **WYSIWYG Release ##** > **WYSIWYG ArtNet Configuration Utility**.

Note: If the WYSIWYG ArtNet Configuration Utility does not appear on the Start menu, locate the executable file in the Bin folder at the path where you installed WYSIWYG (e.g., C:\Program Files\WYSIWYG R27\Bin).

Result: The WYSIWYG ArtNet Configuration window appears:



2 From the drop-down box, select the IP address at which the ArtNet data is being received by this computer.

3 Click *OK* to save your changes and close the utility.

4 Run WYSIWYG and start a new file.

5 Insert a venue, some hang structures, and fixtures.

6 Patch the fixtures to one or more DMX Patch Universes.

Note: Patching to EDMX will not work.

7 Switch to LIVE mode.

8 From the Live menu, select **Device Manager.**

9 In the Device Manager window, click *New*.

10 In the Library Selection window, search for Art-Net (note spelling).

11 Select the single search result and click *Insert*.

12 With Artistic License Art-Net selected in the device list, click *Properties*.

13 Bind the Universes that you created in step 6 to the incoming ArtNet ports.

14 When the binding is complete, click *OK*.

15 In the Device Manager window, with Artistic License Art-Net selected in the device list, click *Connect*.

16 Close the Device Manager window.

Result: You are now connected to ArtNet and WYSIWYG will begin reacting to incoming ArtNet data.

Appendix E

Connecting to Luminair



This new interface enables you to export Fixture POVs and their associated information to the Luminair app, allowing Luminair users to associate a visual reference with cues in the app. This option is only available with WYSIWYG Design and WYSIWYG Perform. You can also use the Batch Print feature in PRES mode to send PDF files of reports, layouts, and worksheets to the Luminair app.

In this appendix

Connecting to Luminair 581

Connecting to Luminair

Introduction

The Luminair interface enables you to export Fixture POVs and their associated information to the Luminair app, allowing Luminair users to associate a visual reference with cues in the app.

To use it, you must ensure that there is an accessible WIFI network for the computer that is running WYSIWYG and for the iPhone/iPod Touch that is running the Luminair 2.0 app.

Note: This option is only available with WYSIWYG Design and WYSIWYG Perform.

To connect to Luminair

1. In WYSIWYG, click **Options** > **Application Options**, and then click the *Additional Interfaces* tab.
2. Click the checkbox to enable the Luminair interface. (If this option was previously turned off, please save your file and then stop/restart WYSIWYG.)
3. In the Luminair app, on the *Main* tab, go to *Preferences* and then scroll to the bottom and select *Bonjour File Transfer*.
4. Set a four-digit password, and then click *Start Server*. Once this completes successfully, the IP address of this device will appear in the lower part of the screen.
5. Restart WYSIWYG.
6. Click **Options** > **Application Options**, and then click the *Additional Interfaces* tab.
7. In the Server IP Address box, specify the IP address of the iPhone/iPod Touch device where Luminair is running.
8. In the Server Password box, type the same password as that which you set in the Bonjour File Transfer window in Luminair.
9. Click *OK*.
10. In your WYSIWYG file, insert some fixtures that you would like to upload to Luminair.
11. In CAD Mode, click the Quad tab.
12. In any of the CAD Wireframe views, select the fixtures that you want to export, and then right-click in the Shaded View and select **Export Fixture POVs**.
13. Here you have the option to render and upload the selected fixtures' POVs to a folder or to Luminair. Select *Upload to Luminair*, and then click *Export*. The Export process begins and the Fixture POVs will flash across your screen as they are exported/loaded into Luminair.
14. Once the upload is completed, click *Stop Server* to stop the File Transfer server on the iPhone/iPod Touch.
15. Click *OK* to confirm that you are closing the connection.
16. In Luminair, navigate to the Controls tab.
17. In the top left corner, use the '+' button to add cues, and to import the quick cues from WYSIWYG.

18 Click *From WYSIWYG*.

Result: A list of all wysiwyg files previously used to upload POV images is displayed.

19 Select the file name that contains the POV images that you would like to open.

Result: When loading completes, the upload POV images have created new cues (identified with a 'w' in the top left corner). Each of these cues now has an associated image and fixture information that was exported from WYSIWYG.

20 To view details associated with a cue, click on the white label under the number. This will open the properties of this cue, displaying its fixture information (imported from WYSIWYG). The image is also assigned and its thumbnail is visible in the top left corner. Click on this thumbnail to enlarge the image.

21 Once you are finished using WYSIWYG and Luminair, you can disable the Luminair Interface from the Application Options > Additional Interfaces tab.

Viewing PDF files of worksheets, layouts and reports

You can use the Batch Print feature in WYSIWYG's PRES mode to send PDF files of worksheets, layouts, and reports to the Luminair app. For details on using the Batch Print feature, see “Batch printing” on page 462.

Appendix F

Using the 3D Connexion Mouse with WYSIWYG



This new interface enables you to use a 3D Connexion mouse device, making camera movement and design in all six Degrees of Freedom (DoF) simpler and more intuitive. The device combines mouse interaction with many common buttons, all at your fingertips. Use the 3D Connexion mouse to control camera movement in Shaded Views and adjust the focus (pan/tilt) of a fixture.

In this appendix

Using the 3D Connexion Mouse 585

Using the 3D Connexion Mouse

Introduction

This new interface enables you to use a 3D Connexion mouse device, making camera movement and design in all six Degrees of Freedom (DoF) simpler and more intuitive. The device combines mouse interaction with many common buttons, all at your fingertips. Use the 3D Connexion mouse to control camera movement in Shaded Views and adjust the focus (pan/tilt) of a fixture.

3D Connexion Menu

You can display the 3D Connexion menu at any time by pressing the *Menu* button while in the Shaded view.

- **Speed**: You may modify the speed and sensitivity of the input from the 3D connexion mouse to any one of Low, Medium, and High.
- **Motion Control – Pan/Zoom Only**: Select this option to limit the input to control only Pan and Zoom while in Shaded views.
- **Motion Control – Rotate Only**: Select this option to limit the input to control only rotation while in Shaded views.
- **Motion Control – Dominant Motion**: Select this option to tell WYSIWYG to only consider the dominant motion while using the mouse. This is useful if you want smoother control over the Camera.
- **Navigation Mode – Move Camera**: Select this option to allow you to move the camera in Shaded views.
- **Navigation Mode – Modify Fixture Pan/Tilt**: Select this option to let you focus selected fixtures without having to hold down the ALT key.
- **Navigation Mode – Encircle Target on Pan**: Select this option to change the way in which you Pan In.

Camera Navigation in Shaded View

The following will be the expected output for the indicated input on the 3D mouse:

- **Push Forward** – Camera moves towards the object/scene.
- **Pull Backwards** – Camera moves away from the object / scene.
- **Pull Up** – Camera moves up while looking at the target.
- **Push Down** – Camera moves down while looking at the target.

The following will be the expected output for input rotation:

- **Tilt Forward** – Camera looks down (Target moves down).
- **Tilt Backward** – Camera looks up (Target moves up).
- **Twist Right (Clockwise)** – Camera looks to the right.
- **Twist Left (Anti Clockwise)** – Camera looks to the left.

Switching the camera position on the 3D Mouse keys

The following occurs when you press the 3D Mouse keys on the SpacePilot Pro model:

- **Fit** - Automatically positions the camera forward or backward to a point where the whole scene is visible. Camera to target distance is maintained.

- **T** – Target is set to the center of the scene and camera is positioned to be looking at the Top View.
- **B** – Target is set to the center of the scene and camera is positioned to be looking at the Bottom View.
- **F** – Target is set to the center of the scene and camera is positioned to be looking at the Front View.
- **BK** – Target is set to the center of the scene and camera is positioned to be looking at the Back View.
- **R** – Target is set to the center of the scene and camera is positioned to be looking at the Right View.
- **L** – Target is set to the center of the scene and camera is positioned to be looking at the Left View.
- **Iso1** – Target is set to the center of the scene and camera is positioned to be looking at the First Isometric View (Camera at the top left when looking from the front).
- **Iso2** – Target is set to the center of the scene and camera is positioned to be looking at the Second Isometric View (Camera at the top right when looking from the front).
- **Translation or Rotation Only** - If you only want to translate or rotate the camera. The undesired motion will be filtered out. The Translations and Rotations toggle buttons will achieve this effect. In Space Pilot Explorer, use the 1 and 2 keys to achieve this.
- In the Space Pilot Explorer model, set the B (Bottom) and BK (Back) views by pressing and holding the *SHIFT* key while pressing the T or F keys accordingly. Set Iso1 and Iso2 by pressing 1 and 2 while pressing and holding the *SHIFT* key.

Focusing Pan / Tilt of Fixtures

Limited fixture control in Shaded view can be achieved using the 3D Connexion mouse:

- With only a single fixture selected, press the *FIT* key and the camera will be positioned behind the fixture looking toward the target on which the fixture is focused.
- With any number of fixtures selected, press and hold the *ALT* key while tilting the 3D mouse forward and back or twisting it left and right. The fixture pan and tilt adjusts accordingly. Note that fixture control follows the WYSIWYG control conventions. Only conventional fixtures may be focused in CAD mode, and only moving head and mirror fixtures may be focused in DESIGN mode.

Appendix G

Using the WYSIWYG Dongle Update Utility



The WYSIWYG Dongle Update Utility can be downloaded from the WYSIWYG Members website, https://members.wysiwygsuite.com/. Using this tool, you can view and track license distribution information for the dongles on your network.

In this appendix

WYSIWYG Dongle Update Utility 589

WYSIWYG Dongle Update Utility

Updating your dongle

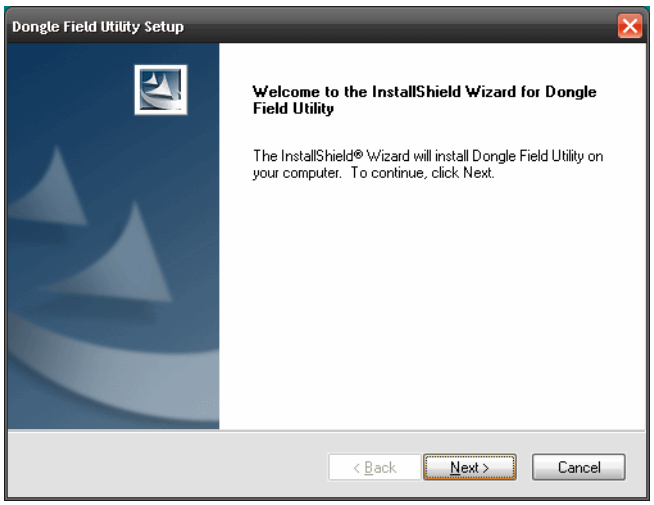
Your dongle is not only the key to open your WYSIWYG software, but it also has the security protocol (to protect your investment) and other information imbedded on it that establishes you as the registered licensee. From time to time, CAST will update its security protocol to protect its intellectual property and the investment of its End-Users. Consistent with the new security protocol introduced in the release of WYSIWYG R23, it is time to test all dongles to ascertain they meet the new technical specifications of this security update, and update the dongles to run this and future releases.

Accordingly, all dongles that have not been tested for, or have ever run, Release 23 or later must be tested to check whether they meet the current technical specifications required by our security protocol. Your dongle will be polled and information sent to CAST to validate its compatibility with the new security protocol. Dongles which pass this test can be updated using an Update Code which they will receive in an email.

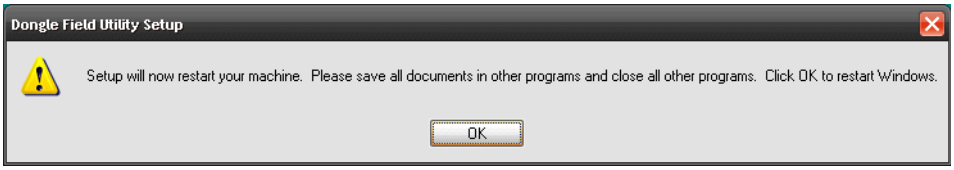
Conversely, dongles that fail this test cannot be updated using this utility, and instead the dongle must be replaced with a new one by CAST Software. If the results of this test reveal that your dongle cannot run WYSIWYG Release 23 or later, contact CAST Software. You will receive instructions on how to receive a new dongle, and once the new replacement dongle arrives, use the WYSIWYG Dongle Replacement Utility to transfer the license from your current dongle to the new dongle. (Refer to AppendixH, “Using the WYSIWYG Dongle Replacement Utility”). If your WYSIWYG Membership is in good standing, then all dongle exchanges are free of charge.

To test and update your dongle

1 Download and install the WYSIWYG Dongle Update Utility by accessing the secure WYSIWYG Members website, https://members.wysiwygsuite.com. This package includes the WYSIWYG Dongle Update Utility for determining whether or not your dongle needs updating, and whether or not you require Microsoft .NET Framework 3.5. It also includes Safenet Sentinel Drivers 7.5.0, which are required by the WYSIWYG Dongle Update Utility.



2 When the installation is complete, you are prompted to close all other programs running, and restart the computer.



3 Insert your dongle into a USB port on your computer.

4 Ensure that you are connected to the Internet.

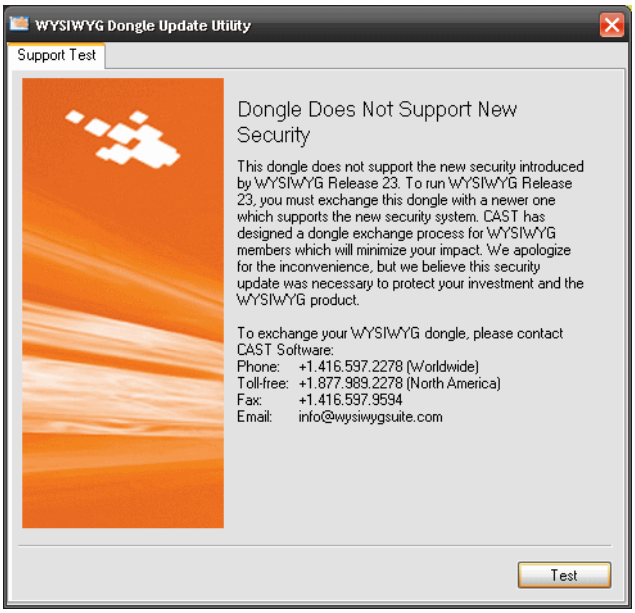
5 Locate DongleUpdateUtility.exe on your computer and double-click it to start the utility. (The Dongle Update Utility files are installed in C:\Program Files\WYSIWYGDongleFieldUtility\Bin.)

The WYSIWYG Dongle Field Utility dialog opens.

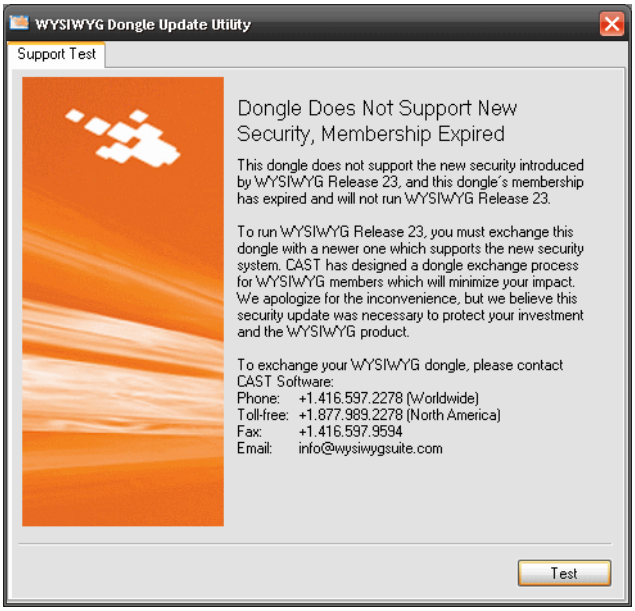


6 Click the *Test* button.

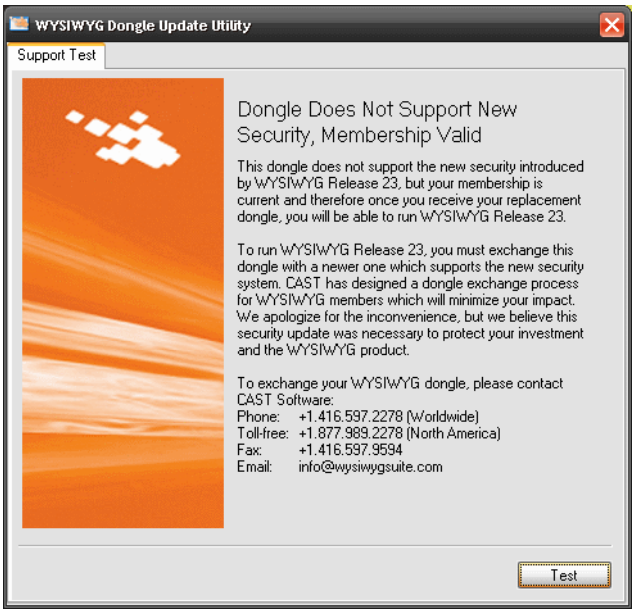
a. If your USB dongle predates 2005 or is a Parallel dongle which cannot be read by the Utility, the following general message appears.



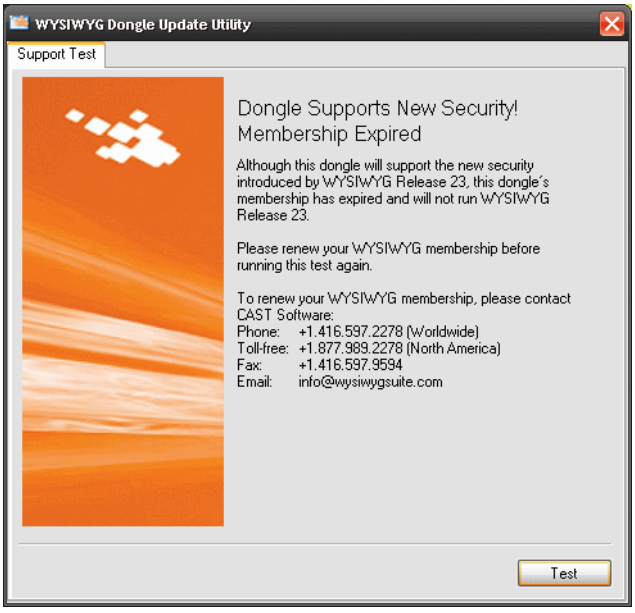
b. If your dongle is not supported by WYSIWYG R23 or later and membership is expired, the following message appears.



c. If your dongle is not supported by WYSIWYG R23 or later and you are a Member in good standing (meaning your Membership is current), the following message appears.

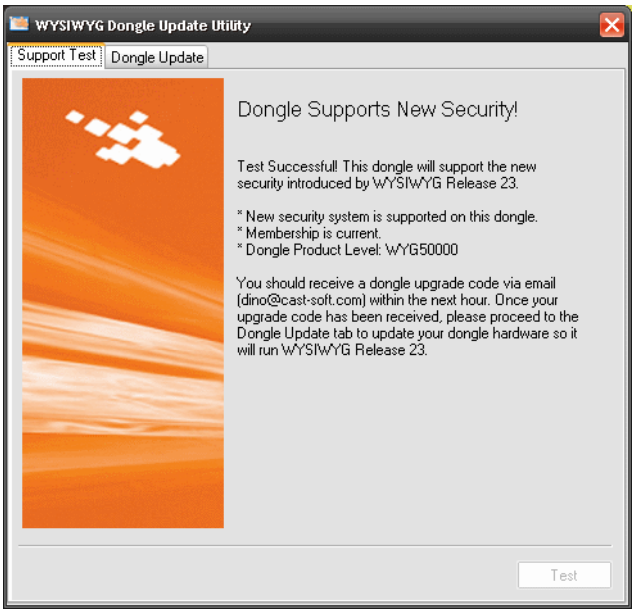


d. If your dongle is supported by WYSIWYG R23 but your Membership has expired, the following message appears.

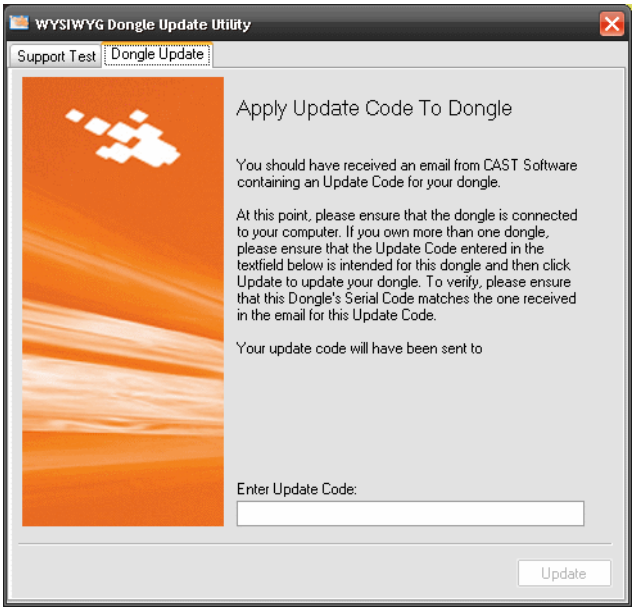


Contact CAST Software to update your membership.

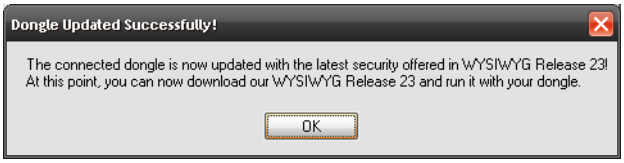
e. If your dongle is supported by WYSIWYG R23 and your membership is in good standing, the following message appears.



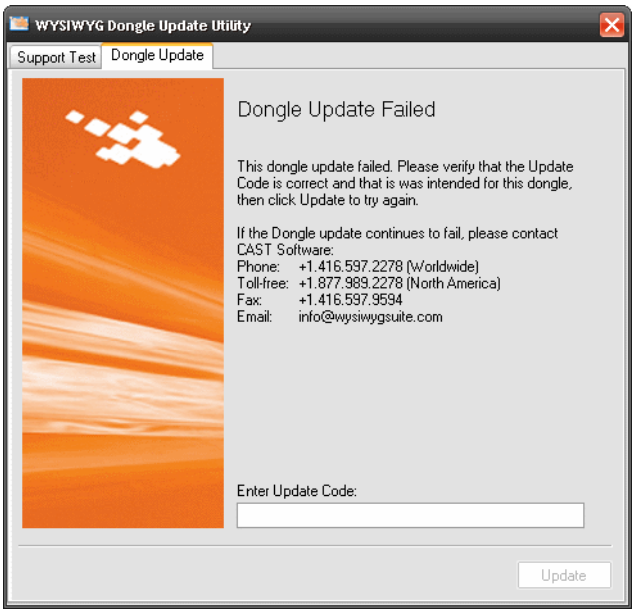
CAST Software will email you an Update Code. Copy the Update Code from the email and paste it in the text box at the bottom of the Dongle Update tab.



You can now use your updated dongle to run WYSIWYG R23 software.



If an error occurs during the update step, the following message appears.



Appendix H

Using the WYSIWYG Dongle Replacement Utility



If your dongle cannot be updated, you must replace it. Once your replacement dongle arrives, use the WYSIWYG Dongle Replacement Utility to transfer the license from your current dongle to the new dongle. If your WYSIWYG Membership is in good standing, then all dongle exchanges are free of charge.

In this appendix

WYSIWYG Dongle Replacement Utility 597

WYSIWYG Dongle Replacement Utility

Replacing your dongle

Use the WYSIWYG Dongle Replacement Utility to transfer the license from your current dongle to your new replacement dongle.

To transfer your license to your replacement dongle

1 Download and install the WYSIWYG Dongle Replacement Utility.

This package includes the WYSIWYG Dongle Replacement Utility which transfers your WYSIWYG license information from your current dongle to a new dongle that supports the new security system. If necessary, the installer also installs Microsoft .NET Framework 3.5 (if you previously installed the WYSIWYG Dongle Update Utility, this step is skipped). Finally, the installer includes the Safenet Sentinel Drivers 7.5.0, which are required by the WYSIWYG Dongle Replacement Utility.

2 When the installation is complete, you are prompted to close all other programs running, and restart the computer.

3 Insert your original dongle into a USB port on your computer. Ensure that this is the only dongle currently connected to the computer.

4 Ensure that you are connected to the Internet.

5 Start the utility. **Read the disclaimer carefully**, and if the conditions apply, click *OK* to continue.

The conditions under which you should run this utility are the following:

a. You have already tested your dongle with the WYSIWYG Dongle Update Utility, which reported that this dongle cannot run WYSIWYG Release 23.

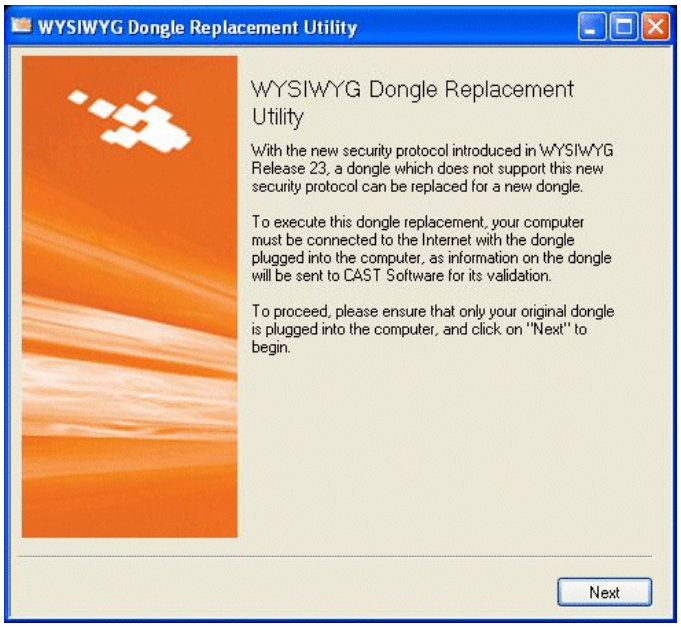
b. You contacted CAST Software to notify them that your dongle needs to be replaced and a new replacement dongle was shipped to you.

c. You currently have both your original and replacement dongle in your possession.

d. You would like to transfer the license from your original dongle to your replacement dongle.

e. You have downloaded this utility from the WYSIWYG Member's website, https://members.wysiwygsuite.com/.

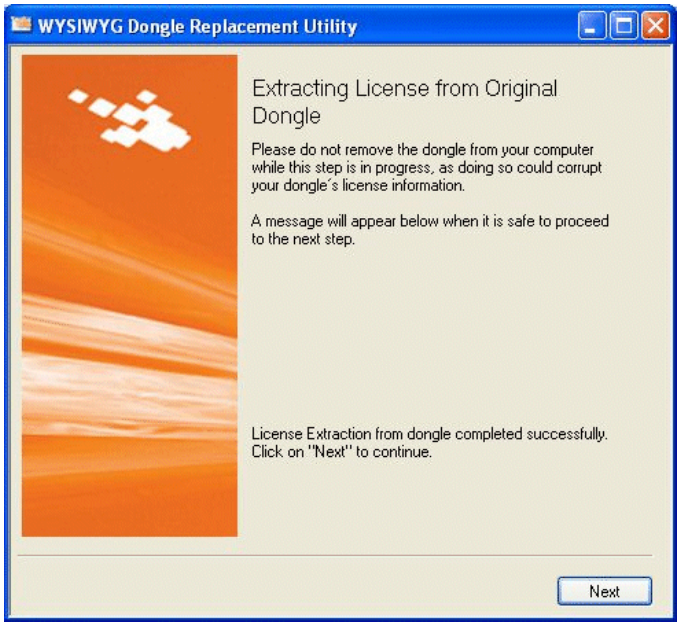
The following dialog appears.



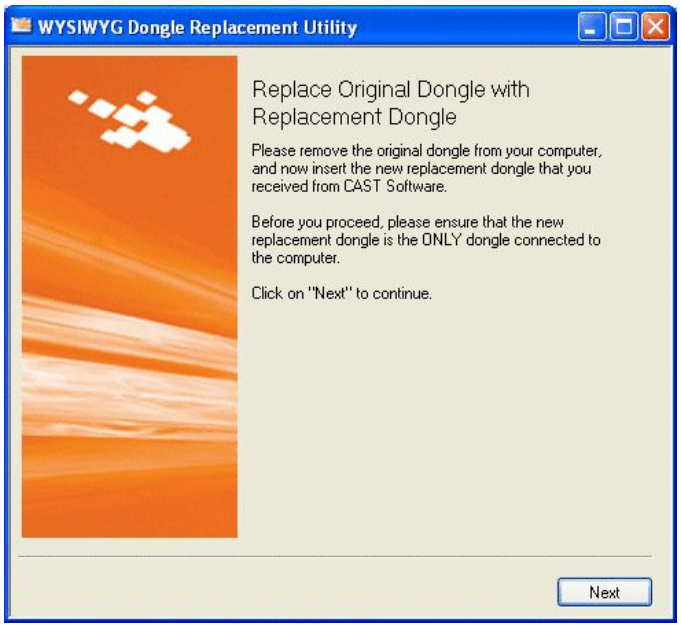
6 With your original dongle connected to the computer, click *Next* to proceed with the installation.

During this step, the utility is extracting. Do not remove the dongle.

Once completed, instructions appear at the bottom of the dialog to proceed to next step.

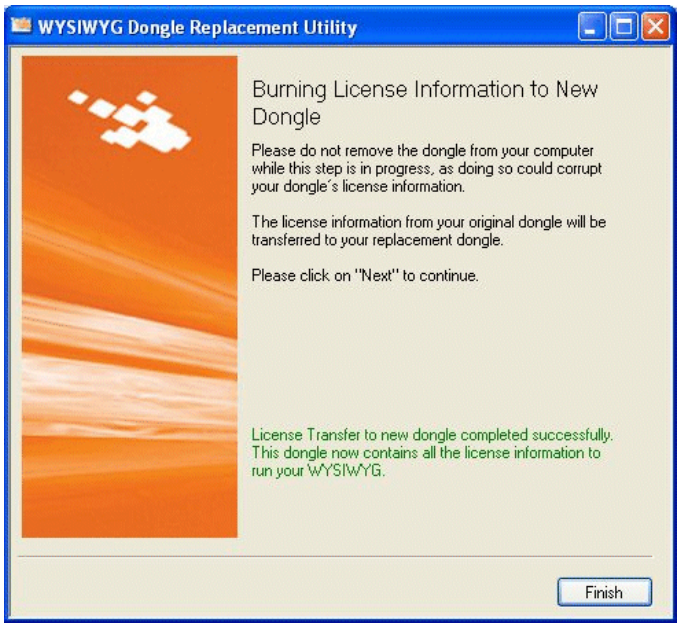


7 Click *Next*.



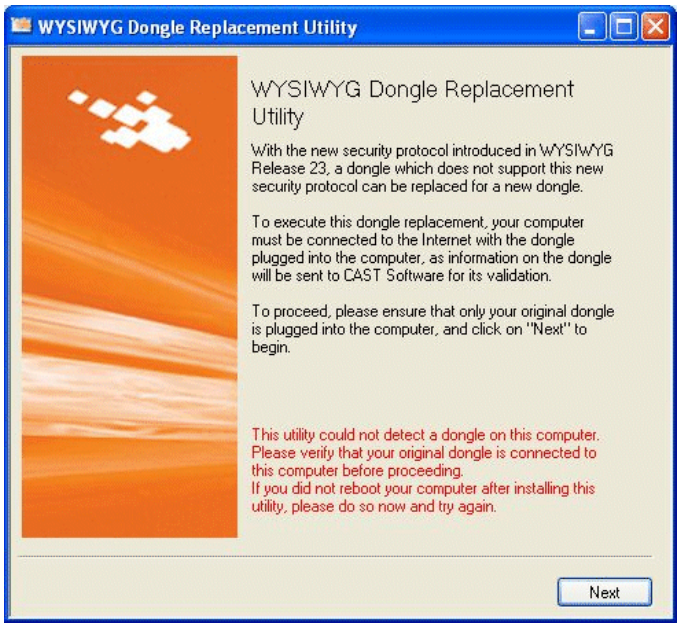
8 Remove the original dongle, insert the new dongle, and then click *Next*. The utility starts transferring the license to the new dongle.

9 Once the transfer has completed successfully, a message appears. The new dongle should be functional and the original dongle will no longer run WYSIWYG

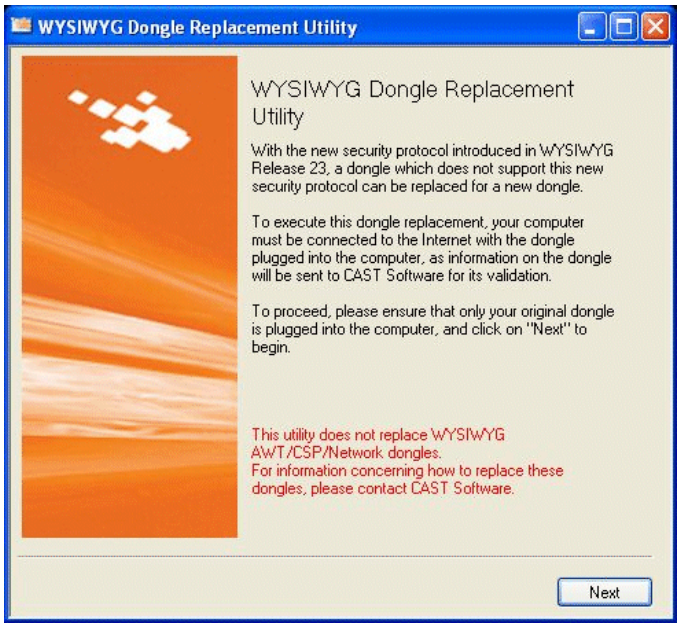


Notes

- If the utility could not find the dongle, try restarting your computer and running the utility again.

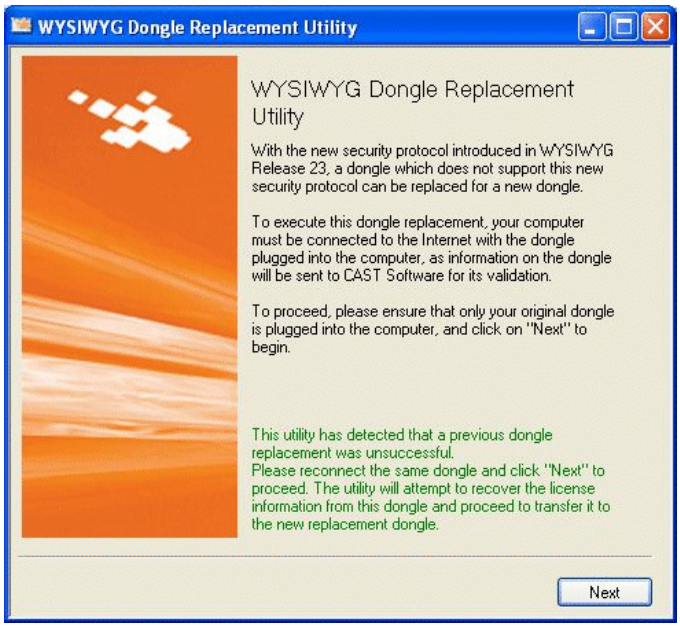


- If you inserted a CSP/AWT/Network dongle, the following message will appear.



These dongles do not work with this utility. Contact CAST Software for information.

- If a replacement attempt was not completed successfully, the utility will run in recovery mode.



This means that the previous attempt of transferring the WYSIWYG license from one dongle to another failed.

The utility will prompt you to insert the dongle used during the failed attempt.

Index

Numerics

3D Connexion Mouse
Using, 585

A

Absolute coordinates, 69, 191
Accessory, 315
Activation, 38
Add Insertable command, 384
Adding
floorplans, 127
Additional Interfaces tab, 393
Aiming fixtures, 281
Alias definition, 297
Aligning objects, 235, 431
Ambient light, 84
Antialiasing options, 505
Aperture, 500
Appearance tab
renaming elements in, 199
Application Options, 389
Arc Length Dimension tab, 209
Arc Length Dimensions, 150
Arcs
Drawing in CAD mode, 146
Drawing in New Plots, 428
Arena, 142
Arranging shortcuts, 57
Array, 236
Grid pattern, 238
Using to hang fixtures, 275
Aspect ratio, 451
Assembly Snap, 106, 268
Atmospheric options, 501
Attribute layout, 426
Attributes (Data fields), 358
Auto load, 390
Auto save, 390
Autofocus, 544, 571
AutoPatch, 539
AutoUnit, 257, 258, 398
Axes
drawing, 170

B

Back view, 104
Background Rendering Manager
About, 508
Buttons, 513
Configuring, 508
Deleting jobs from, 516
Exporting images from, 515
Moving dequeued jobs back into queue, 515
Moving jobs to top of queue, 515
Moving jobs up or down in queue, 515
Pausing, 514
Removing jobs from queue, 515
Resuming, 514
Taskbar icons, 511
Backup files, 390
Barndoors, 315
Batch printing, 462
Baud rate, 394
Beam angle, 299, 543
Beams, 406
Big Layouts, 436
Bind
to console, 526
Bitmaps, 463
Black Box, 142
Blocks
Custom library items, 175
Groups, 191
Boolean Operations
and intersection, 242
and projection, 242
and view planes, 242
Boolean operations, 241
Booms, 263
Borders and legs, 162
Break tool, 227
Bringing to Front, 193
BTX, 381
Budget
Cost, 297, 359
Bulb
Lamp, 299, 359

C

CAD environment, 98
CAD items, 445
CAD Options toolbar, 60
Camera, 164
changing the hotkey number, 166
deleting the hotkey number, 166
locking in shaded view, 166
using hotkeys, 165
Using Pitch, Roll and Yaw, 167
Camera controls, 78
Camera path editor, 494
Camera Paths, 493
Camera paths
patching, 550
Camera toolbar
using, 165
Center line, 145
Channel count, 68
Channels
Simulation, 68
Circle, 146
Circles, 427

Circuit Name data field, 358
Circuit Number data field, 358
Circuit Type data field, 359
Circus tents, 143
Close the show document, 386
Color
  Inserting, 316
  Inserting multiple, 362
Color data field, 358
Color Temperature and Intensity Falloff features, 92
Color temperature calculations, 92
Color wheel, 478
Column Width Size, 375
Columns in Data mode, 358
ComEdit toolbar, 61
Compass, 170
Cones
  drawing, 161
Connecting, 526
  console, 526
Console data field, 359
Consoles
  connecting, 526
Consolidated Mesh feature, 212
  using, 252
Content properties, 444
Convert
  Lines into surfaces, 229
Convert tool, 228
Co-ordinate display, 390
Coordinate notation, 98
Coordinates, 69, 98
Copy, 429
Cost data field, 359
Creating a new DMX universe, 367
Creating a new spreadsheet, 365
Cross-fading between lighting looks, 487
Cuelists, 490
Current
  Set current layer, 218
Curved pipes, 260
Custom, 319
Custom color scroll, 317
Custom color wheels, 317
Custom fixtures
  creating, 278
  deleting, 279
Custom gobo wheels, 317
Custom gobos, 319
  deleting, 320
Custom library items, 136
  creating, 176
  deleting, 176
Custom settings, 389
Customer Experience Improvement Program (CEIP), 30
Customizing spreadsheet views, 359
Cut, 429
Cuts and Adjustments tool, 289
Cycle Selection, 186
Cylinders, 159

D

Data error tab, 400
Data fields, 358
Data filters, 362
Data toolbar, 61
Decimal display, 390
Delete, 430
Design toolbar, 61
Design tools, 474
Designer's Remote, 546
Device Manager, 526
Device Manager toolbar, 61
Dimension
  Arc Length, 150
  Linear, 150
  Radial, 151
Dimension line, 150
  fonts used in, 150
Dimmer data field, 358
Distance tool, 241
Distribute Objects, 240
Distributing fixtures, 276
Divide tool, 231
DMX, 526, 534
Document summary, 395
Dolly, 79
Dongle reauthorization, 35
Dot line, 145
Draw defaults, 404
Draw menu
  Wall command, 154
Draw toolbar, 62
Drawing
  motion axes, 170
  Walls, 154
Drawing aids, 138
Drawing pipes, 259
Drawing rectangles, 148
Drawing tips, 138
DWG, 381, 382
DWG/DXF files
  importing, 116
DXF, 381, 382
DXF/DWG files, 116

E

Edit (Pres) toolbar, 63
Edit toolbar, 62
Editing a key, 458
Editing a legend, 454
Editing objects, 185
Editing symbols, 453
EDMX, 393
EDMX address, 304
Elements
  renaming, 199
Enhanced beam quality
  Color Temperature & Footprint Intensity Falloff features of, 92
Entering data in the spreadsheet, 359

Environmental options, 504
Errors, 376
Ethernet, 544
Exit WYSIWYG, 388
Export to DWG/DXF, 124
Exporting
  to 2D DWG/DXF, 124, 125
  to 3D DWG/DXF, 125
Extrude, 233

F

Fading between lighting looks, 487
fastFan
  Using in CAD mode, 286
Field angle, 299
File menu, 381
Fillet, 234
Filter
  Data filter, 362
Find tool, 360
Fixture Details tab, 422
Fixture layout, 308
Fixture notation, 308
Fixture properties
  Photometrics, 299
Fixture snap, 106
Fixtures, 274
  Distribute and array feature, 277
  Distributing along pipes/truss, 276
  Distributing between two points on pipes/truss, 277
  Distributing from the center of pipes/truss, 277
  Distributing in a direction on pipes/truss, 277
  Hanging multiple, 275
  Inserting in CAD mode, 274
  Selecting, 276
  Using Array feature to hang, 275
Flight Case, 351
Floor mount, 257, 263
Floorplans
  adding, 127
  Hiding after importing, 132
Flow control, 394
Focus
  Design tool, 479
Focus data field, 358
focus lines
  assigning fixtures to, 284
Focus position, 314
Focus tool, 479
Focusing
  click and drag method, 480
Focusing fixtures, 281
Font
  changing in New Plots, 428
  changing in Text Labels, 149
Footnotes data field, 359
Footprint Intensity Falloff, 93
Footprint tool
  using (CAD mode), 293
  using (Design mode), 482
Frame size, 315
Frame tents, 143
French curve, 145
Front view, 104, 105
Full screen mode, 52

G

Gels, 477
  Inserting, 316
  Inserting multiple, 362
General object properties, 195
Gobo data field, 358
Gobo wheels
  Custom, 317
Gobos
  Custom, 319
  Design tool, 477
  Inserting, 316
Graphics, 463
Grid array, 238
Grid options, 404
Group tab, 191
Grouping, 191

H

Hang structure, 257, 281
Hanging, 274
Hanging fixtures, 274
Hanging multiple fixtures, 275
Hanging truss, 263
Hatch style, 314
Hatching style, 156
Hidden line, 145
Hide Floorplan, 132
HogPC
  Connecting Autofocus to, 571
Hookup data field, 358
Hot keys, 562
  Patch views, 568
  Presentation views, 566, 567
  Simulation, 564

I

Image Console
  configuring, 535
  controlling LES grids with, 538
Image items, 450
Image Manager, 343
Images, 463
Imperial units, 99
Importing
  DWG/DXF files, 116
  SketchUp files, 107, 108, 112
Importing DWG/DXF files, 118, 121
In-Place update, 23

Insert (Pres) toolbar, 63
Inserting
  Color in Data mode, 362
  Fixtures in CAD mode, 274
  Fixtures in Data mode, 361
  Focus positions in Data mode, 361
  Positions in Data mode, 362
Installing
  In-Place update, 23
  WYSIWYG, 22
  WYSIWYG Learn, 24
  WYSIWYG Network, 24
Installing WYSIWYG, 23
Intensity design tool, 475
Interactive mode, 107, 402
Intersect Objects, 241
Intersecting objects, 248
Intersection
  and the Boolean Operations feature, 242
Intruction Tooltip, 69
Inverse Square Law calculations, 93
Isometric view, 104
Item properties, 444

J

jpeg file, 320, 438, 463, 501

K

Key
  Instrument labels, 308
Keyboard and mouse control, 71
Keyboard shortcuts, 561
Keys, 456

L

Ladders, 257, 263
Lamp Type data field, 359
Lamps, 134
Latitude, 401
Layer data field, 359
Layers, 218
Layers toolbar, 63
Layout
  fixture data, 308
Layout tabs
  CAD, 97
  Data, 357
  Design, 473
  Live, 525
  Pres, 413, 519
Layouts
  locking items in, 460
  Locking out items in, 460
  unlocking items in, 460
Layouts (Pres mode), 436
Lease
  Renewing, 42
Lease Details
  Obtaining, 41
Leases, 39
LED Wizard, 322
Left view, 104
Legend items, 452
Library Browser, 133
  using the Search feature in, 134
Library objects
  applying textures to, 136
  replacing, 135
Library shortcuts, 56
Light emission, 199
Line
  Drawing lines, 144
Line color, 222
Line weight, 222
Linear axis, 171
Linear Scales, 152
Lines, 428
  Converting into surfaces, 229
Live toolbar, 64
Location setup, 170, 401
Longitude, 401
Looks
  and moving scenery, 488

M

Marker grips, 194
Masking, 162
Material Reflections, 216
Materials, 198, 213
Measurement mode, 150
Membership and Leases, 39
Menu bar, 59
Merge, 385
Merge Objects, 241
Merging
  SketchUp files, 112
Merging documents, 385
Metric units, 99
Micro Layouts, 436
MIDI, 544
Mirror tool, 194
Mirroring, 194
Missing coordinate, 102
Model data field, 359
Modes, 49
Modifying wireframe views, 73
Motion axes
  drawing, 170
Motion axis
  adding vertex to linear, 173
  attaching objects to, 172
  attaching to a DMX patch, 172
  attaching to a motion patch, 172
  changing path of, 173
  drawing linear, 171
  drawing rotation, 171
  moving objects on with a console

device, 532
moving objects on with a motion control system, 532
patching to DMX console device, 370
patching to motion control system, 371
Motion control system
moving objects with, 532
Mountable yokes, 315
Move, 430
Move Page, 422
Move tool, 191
Moving head, 304
Moving mirror, 304
Moving scenery
reading the patch for, 373
viewing, 488, 489
Moving Scenery tool, 482
using, 482
Multi-source light
applying color to with Quick Tools, 316

N

Navigation shortcuts, 55
New Back Elevation Plot, 419
New Front Elevation Plot, 419
New Left Section Plot, 419
New Plan Plot, 418
New Plot View Properties tab, 420
New Plots, 418
Attribute layout, 426
changing font used in, 428
Creating a new plot type, 418
Drawing objects, 427
Editing objects, 429
Inserting into layouts, 443, 450
Modifying properties of a plot, 419
Objects, 422
Objects drawn in New Plots, 432
Printing, 434
Reestablish links, 423
Relationship with objects in CAD, 423
Relationship with objects in Data, 423
Selecting objects, 429
Updating, 419
Working in, 418
New Right Section Plot, 419
New Show, 381
Nudging, 192
Number of color frames data field, 359
Number of data channels data field, 359
Number of lamps data field, 359

O

Object representation in New Plots, 422
Object-specific properties, 204
Offset, 263
Offset data field, 359
Open a show document, 381
Open GL tab, 392
Options, 389
Orbit, 78
Origin, 98, 404
Ortho, 69, 105
Owner data field, 359

P

Palette shortcuts, 486
Pan data field, 359
Paperwork, 414
Passwords, 44
Patch, 366
changing width of cells in, 375
Patch data field, 358
Patch universe, 366
Patch views
Hot keys, 568
Patching
motion axes (DMX), 370
motion axes (motion control system), 371
Patching fixtures, 367
Photo quality pictures, 497
Photometrics, 274, 299
Pipe, 162, 279
Pipe and Drape Wizard, 177
Pipe snap, 106
Pipe tape, 467
Pipes
Distributing multiple fixtures on, 277
Pivot, 79
Plan view, 104, 105
Plot type, 104
Points, 145
Polygons
drawing, 153
Popup frames, 51
Ports
console, 526
Position, 257
Position data field, 358
Position Manager, 257
Precision, 208, 209, 210, 404
Presence option, 501
Presentation items, 438
Presentation views
Hot keys, 566, 567
Print, 386
Printing
Batch, 462
Printing layouts, 462
Product Configuration Wizard, 31
Production Team Info Wizard, 469
Projection
and the Boolean Operations feature, 242
Properties
Dimension, 208
Errors, 378
Focus position, 282
General, 195

Layouts, 437
Legend, 455, 459
Library item, 136
Line, 206
Patch universe, 373
Pipe, 262
Point, 206
Report, 414
Spreadsheet, 363
Symbol, 314
Text label, 207
Wireframe view, 73
Proscenium Arch, 141
Protractor tab, 210
Purge command, 384
Purpose data field, 358

Q

Quick focus, 285
Quick Light Emission Tools, 201
Quick tools, 299

R

Radial Dimension tab, 209
Radial Dimensions, 151
Radiosity, 503
Reading the patch, 372
Reauthorization
for dongles, 35
Redesigning headings, 417
Redo, 190
Regional profile, 390
Regional settings, 401
Relative, 191
Relative coordinates, 69
Rename fixture type, 297
Renaming object elements, 199
Render shortcuts, 506
Rendering, 354, 497, 547
Replace tool, 360
Replacing fixtures, 278
Reports, 414, 522
Creating, 416
Reset origin, 98
Resizing
Spheres, 194
Revolve, 253
Right view, 104
Riser, 153
Roll, 80
Room Builder, 138
Rotate, 431
Rotate tool, 193
Rotating, 193
Rotation axis, 171
Ruler
changing origin of, 101
toggling on and off, 101
Ruler icons, 100
Ruler Tool, 100

S

Saturation, 505
Save, 382
Save As, 382
Saving a lighting look, 487
Scenes, 223
Scenes toolbar, 65
Schedules, 414
Screen layout, 49
Scrollbars, 76
Scrollers, 319, 477
Scrolls, 317
Selecting alternate mount points during assembly, 268
Selecting objects, 185
Selection sets, 241
Send the show document through e-mail, 388
Sending to Back, 193
Setting display size for shortcuts, 57
Settings
regional, 401
Shaded View Selection, 187
Shaded views, 78
Shortcut, 55
Shortcut bar
Create Shortcuts
Fixture groups, 484
Images, 463
Layouts, 436
Library Objects, 135
Lighting looks, 487
Palette, 486
Patch Universes, 367
Render, 506
Spreadsheets, 365
Shortcuts, 55
arranging, 57
delete, 56
new, 56
ordering, 56
setting display size, 57
Show details, 405
Show information, 418
Shutter cuts, 288
Shutters tab, 304
Side view, 105
Sidedness
Changing for an object, 203
Sidedness tab, 202
Simulation
Channels, 68
Simulation options, 354, 409, 547
Simulation views
Hot keys, 564
SketchUp files
importing, 107, 108, 112
Merging, 112

Snap, 69, 105, 404
Soft goods, 162
Soft shadow, 504
Solid line, 145
Sphere, 160
Spheres
  Resizing, 194
Spin data field, 359
Spline, 145
Split gels, 362
Splitting video, 338
Spot data field, 358
Spot Numbers
  assigning to fixtures, 544
Spreadsheets, 358
Standard toolbar, 65
Status bar, 68
Status data field, 359
Stock, 317
Streaming video
  reading the patch for, 373
Subtract Objects, 241
Subtract tool
  cutting holes in walls with, 250
Subtracting objects, 249
Surfaces, 153
Symbols, 308, 398, 406, 429, 449

T

Tag data field, 359
Tape printer, 467
Template, 383
Tents, 143
Text label, 148, 255, 428
Text Labels
  changing font of, 149
Textures, 212, 391
  applying to library objects, 136
Tile printing, 421, 438
Tile Printing tab, 421
Tilt data field, 359
Time of Day Lighting tab, 90
Time zones, 401
Timeline tool, 495
Title block, 438, 521
Toolbars, 59, 391
Tools
  CAD, 226
  Design, 474
Tools toolbar, 65
Tooltips, 406, 449
Tophats, 315
Troubleshooting, 4
Truss
  Distributing multiple fixtures on, 277
  Hanging truss, 263
  Indicators for assembly, 263
  rolling, 269
  Rolling truss, 268
Truss cross members, 406, 449
Truss Indicators, 406
Truss snap, 106, 268
Type data field, 358

U

Undo, 190
Ungrouping, 191
Unit data field, 358
Unite Objects, 241
Uniting objects, 247, 248
Universe
  Patch, 366
Unused channels, 68
Upgrades, 21, 38
User Data
  Exporting, 53
  Importing, 53
User Data Profile, 53
USITT symbols, 314

V

Vertex, 227, 232
  adding to a linear axis, 173
Vertical hang structures, 424
Vertical justification of a line, 208
Video
  patching in the patch view, 369
  splitting, 338
Video source
  configuring new, 341
  controlling with a console device, 531
Video tool, 480
View - full screen, 52
View Options
  Error, 378
  Flight Case, 353
  Layouts, 437
  Patch universe, 373
  Report, 414
  Shaded views, 82
  Spreadsheet, 363
  Wireframe views, 74
View shortcuts, 72
View toolbar, 66
Viewer, 410
Viewing
  moving scenery, 488, 489
Views/tools shortcuts, 56
Visual camera path builder tool, 494
Visualization Requirements, 21

W

Walls, 154
Wattage data field, 359
Weight data field, 359
Welcome window, 49
Wheels, 319, 477
Wireframe views, 71

Workplane, 104
WYSIWYG
  Design, 10
  Installing, 23
  Perform, 12, 16, 18
  Report, 5, 9
  starting for the first time, 31
WYSIWYG Learn, 16, 24
WYSIWYG Network, 18
WYSIWYG Perform - Console Edition, 5, 14
WYSIWYG Viewer, 410

Y

Yokes
  mountable, 315

Z

Zoom, 80, 474
  Zooming, 72
Zoom tools, 71