



Quick Start Guide



*Lighting Design Software
Manage, Design and Simulate*



Quick Start Guide

Release 24

July 2009





Contents

| | |
|---|----|
| About this manual | 7 |
| Introduction | 7 |
| Text conventions | 7 |
| Lesson 1 - Understanding the user interface. | 9 |
| Introduction | 9 |
| Step 1 - Starting to work in WYSIWYG | 9 |
| Step 2 - Mode buttons. | 10 |
| Step 3 - Menu bar, toolbars, layout tabs | 10 |
| Step 4 - Shortcut bar | 11 |
| Lesson 2 - Creating a set | 13 |
| Introduction | 13 |
| Step 1 - Inserting a venue | 13 |
| Step 2 - Building your set from CAD objects | 14 |
| Step 3 - Building your set from library items | 14 |
| Step 4 - Building your set using CAD tools. | 15 |
| Lesson 3 - Defining hanging positions | 17 |
| Introduction | 17 |
| Step 1 - Defining a position name. | 17 |
| Step 2 - Drawing a pipe. | 17 |
| Step 3 - Inserting truss | 18 |
| Lesson 4 - Hanging and focusing fixtures | 19 |
| Introduction | 19 |
| Step 1 - Hanging fixtures. | 19 |
| Step 2 - Inserting a focus position | 19 |
| Step 3 - Focusing fixtures using focus positions | 20 |
| Lesson 5 - Assigning fixture attributes | 21 |
| Introduction | 21 |
| Step 1 - Assigning attributes to fixtures. | 21 |
| Step 2 - Filtering and modifying data | 22 |
| Step 3 - Entering sequential numerical data. | 22 |
| Step 4 - Choosing data | 22 |
| Lesson 6 - Building lighting looks in Design mode | 25 |
| Introduction | 25 |
| Step 1 - Creating a new lighting look | 25 |
| Step 2 - Using the Intensity tool | 25 |
| Step 3 - Using the Gobo tool | 26 |
| Step 4 - Using the Focus tool | 26 |
| Step 5 - Using the Color tool | 27 |
| Step 6 - Opening the Render Wizard | 27 |
| Step 7- Cross-fading between lighting looks. | 27 |
| Lesson 7 - Creating and modifying reports | 29 |
| Introduction | 29 |
| Step 1 - Entering show information. | 29 |
| Step 2 - Modifying a report | 30 |
| Step 3 - Formatting report headings | 31 |
| Lesson 8 - Working in the New Plots view | 33 |
| Introduction | 33 |
| Step 1 - Creating a New Plot | 33 |
| Step 2 - Manipulating objects on the plot. | 34 |
| Step 3 - Plotting non-horizontal hang structures. | 34 |
| Step 4 - Inserting objects | 34 |
| Lesson 9 - Creating and modifying layouts | 37 |
| Introduction | 37 |
| Step 1 - Creating a new layout | 37 |
| Step 2 - Inserting objects into the layout. | 38 |
| Step 3 - Modifying CAD items. | 38 |
| Step 4 - Adding legends and keys. | 39 |

| | |
|---|----|
| Lesson 10 - Patching | 41 |
| Introduction | 41 |
| Step 1 - Creating a new patch universe | 41 |
| Step 2 - Patching fixtures and movement axes in the patch layout screen | 41 |
| Step 3 - Repatching a fixture | 42 |
| Step 4 - Reading the patch | 42 |
| Lesson 11 - Inserting and connecting to a console | 45 |
| Introduction | 45 |
| Step 1 - Inserting a console | 45 |
| Step 2 - Connecting to a console | 45 |
| Lesson 12 - Using streaming video | 47 |
| Introduction | 47 |
| Step 1 - Configuring a new video source | 47 |
| Step 2 - Drawing a screen and attaching the video source | 48 |
| Step 3 - Using the Video design tool to play the video | 48 |
| Step 4 - Controlling the video with a console device | 49 |
| Lesson 13 - Using moving scenery | 51 |
| Introduction | 51 |
| Step 1 - Drawing the movement axis | 51 |
| Step 2 - Attaching a movement axis to a patch universe | 52 |
| Step 3 - Attaching objects to the movement axis | 52 |
| Step 4 - Viewing moving scenery | 53 |



About this manual

Introduction

This manual provides an introduction to the basic functionality available in WYSIWYG and is designed to get you started using the program. It is recommended that you use this manual while running the Quick Start Tutorial. The tutorial provides both video and audio elements to help you progress through the 13 lessons. This guide contains a text version of the audio portion of the tutorial, along with accompanying graphics. It also contains some additional information that is not included in the tutorials.

You can access the tutorial from the WYSIWYG Install panel.

Note: To view the video portion of the Quick Start Tutorial, you must have the TechSmith Screen Capture Codec (TSCC) installed on your machine. This codec is available on the WYSIWYG CD for your convenience.

Text conventions

The following text conventions are used in this manual:

- Menus and menu commands appear in **Arial bold**. For example, "from the **Library** menu, choose **Browse Library**."
- 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 the TAB key to enter the missing coordinate."
- References to manuals appear in *italic* font. For example, "for additional information on rendering, refer to the *Reference Guide*."



Lesson 1 - Understanding the user interface

Introduction

In this lesson you will learn about the basic parts of the WYSIWYG user interface.

Step 1 - Starting to work in WYSIWYG

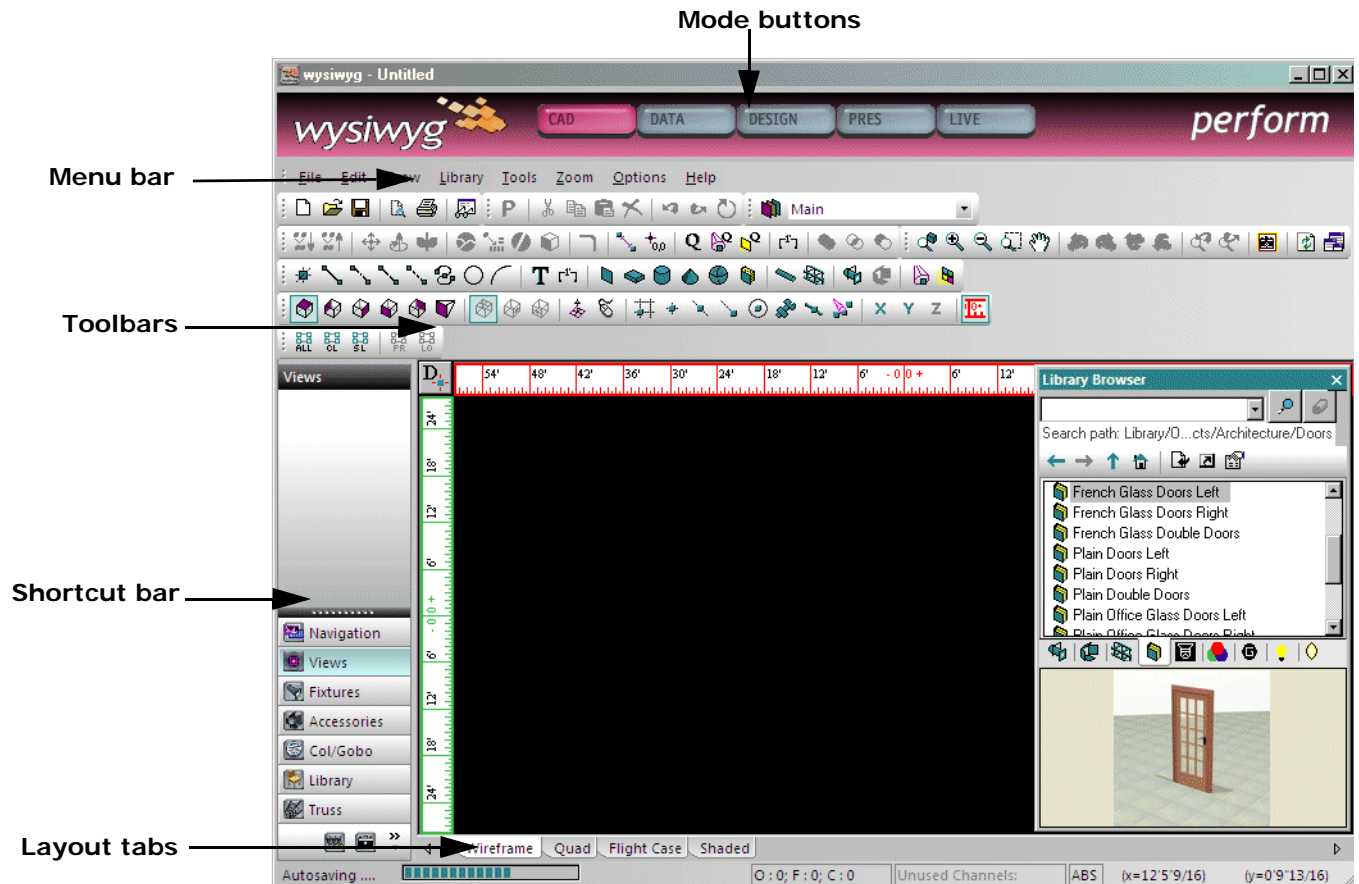
To start working in WYSIWYG

- 1 From the WYSIWYG Welcome window, choose a file to open. 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 files you have recently used are shown on the right.



- 2 For the purposes of this tutorial, click *Empty*.

Result: The WYSIWYG main window is displayed.



Step 2 - Mode buttons

The buttons across the top are the mode buttons. Based on the WYSIWYG product that you have purchased, the mode buttons available are some or all of the following:

- CAD - This is where you will do all of your drawing.
- Data - This is where all of your show information is tabulated on a spreadsheet.
- Design - This mode allows you to turn lights on, compose static lighting looks, and generate renderings.
- Presentation - This is where you create printouts of your show document including reports, plots, and images.
- Live - This is where you pre-cue and visualize your lighting show.

Step 3 - Menu bar, toolbars, layout tabs

Below the mode buttons is the menu bar. The available menus change as needed when moving from mode to mode.

Below the menu bar are the toolbars. Toolbars provide button access to most commands and can be used in lieu of the menu bar.

Located at the bottom of the screen are layout tabs. Each mode has designated layouts. A layout is the configuration of views in your workspace.

Step 4 - Shortcut bar

The shortcut bar is located on the left side of your screen. You can create shortcuts for library items, ease of navigation, and special tools. For example, you can save a navigation shortcut to Live mode/Quad layout so that the next time you want to go back to that location, you can simply click the shortcut.

To create this shortcut

- 1 Select the navigation shortcut bar, right-click, and then select **New Navigation**.
- 2 Type a name for the shortcut and click *OK*.



Lesson 2 - Creating a set

Introduction

In this lesson you will learn how to:

- insert a venue
- build your set using the following methods:
 - from CAD objects
 - from library items
 - using CAD tools

Step 1 - Inserting a venue

Begin in CAD mode. This is the mode in which you create your show drawings. In this example, you add a proscenium arch theatre.

To insert a venue

- 1 From the **Draw** menu, choose **Venue > Proscenium Arch**.

| Stage | | | | | |
|--------------|-------|--------------|-------|--------------|-------|
| Width | 80'0" | Depth | 50'0" | Stage Height | 4'0" |
| Thrust Depth | 8'0" | Thrust Width | 35'0" | Fly Height | 60'0" |

| Arch | | | | | |
|-------|-------|-------|------|--------|-------|
| Width | 40'0" | Depth | 2'0" | Height | 20'0" |

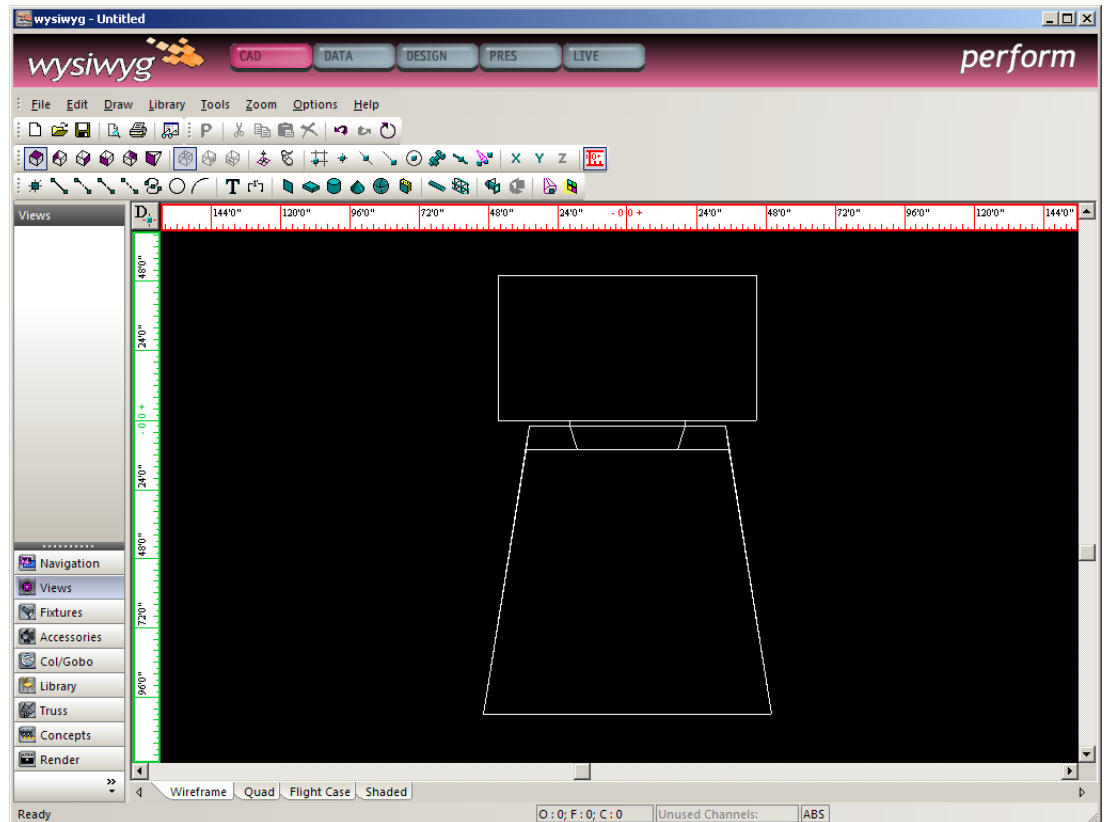
| Auditorium | | | | | |
|------------|--------|-------|--------|--------|-------|
| BOH Width | 100'0" | Depth | 100'0" | Height | 45'0" |
| FDH Width | 68'0" | Slope | 12'0" | | |

Units: Metric Imperial

OK Cancel

- 2 Accept the default values, and then click *OK*.

Result: Your plan view now contains a plan drawing of a proscenium arch theatre.

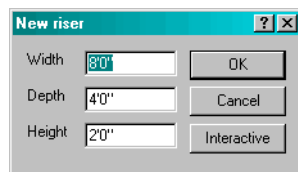


Step 2 - Building your set from CAD objects

A number of CAD objects are available from the **Draw** menu. You can create a set from basic CAD objects such as circles, spheres, cylinders, and risers. In this example, you will insert a riser.

To build your set from CAD objects

- 1 From the **Draw** menu, choose **Riser**.



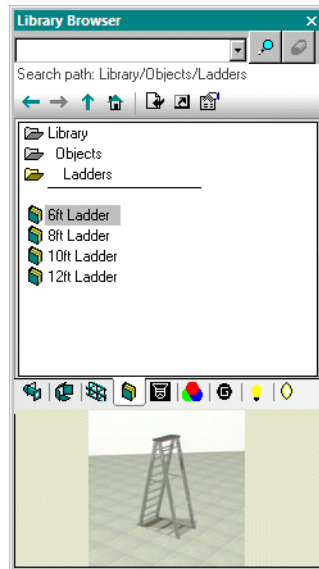
- 2 In the applicable boxes, specify the dimensions of the riser, for example, 8'x8'x2'.
- 3 Click *OK*.
- 4 Click to drop the riser in your drawing.


Step 3 - Building your set from library items

Use the pre-drawn items in the WYSIWYG library to insert objects in the drawing. In this example, you insert a motorcycle.

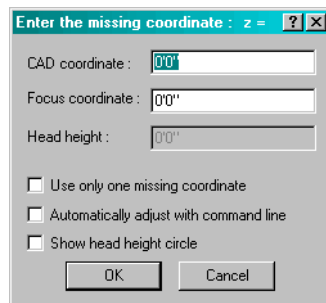
To build your set from library items

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



- 2 Select the *library item* tool  at the bottom of the library browser to ensure that you are browsing the scenic library.
- 3 Navigate to the desired item, and then double-click to insert.

Tip: 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.
- 4 Move the mouse to a point on stage. Before dropping the object in the drawing, press TAB to display the *Enter the missing coordinate* dialog box.



The missing coordinate (X,Y,Z) is the coordinate whose value cannot be entered by clicking on the screen.

- 5 Enter the value for the missing CAD coordinate, for example, 2' (the height of the riser), and then click *OK*.

Tip: As you move your cursor the coordinates are displayed in the bottom right corner of the screen.
- 6 Click to place the object on top of the riser.

Step 4 - Building your set using CAD tools

WYSIWYG is equipped with powerful CAD tools allowing you to draw anything from scratch. In this example, you will draw a basic cyc to illustrate some of these tools.

To draw objects using the CAD tools

- 1 From the *Draw* toolbar, select the item that you want to draw, for example a line.
- 2 To pick the first point of line, type the coordinates (for example, -20,20,0), and then press ENTER.

Note: When you start typing, the ComEdit toolbar is activated at the bottom of the window.

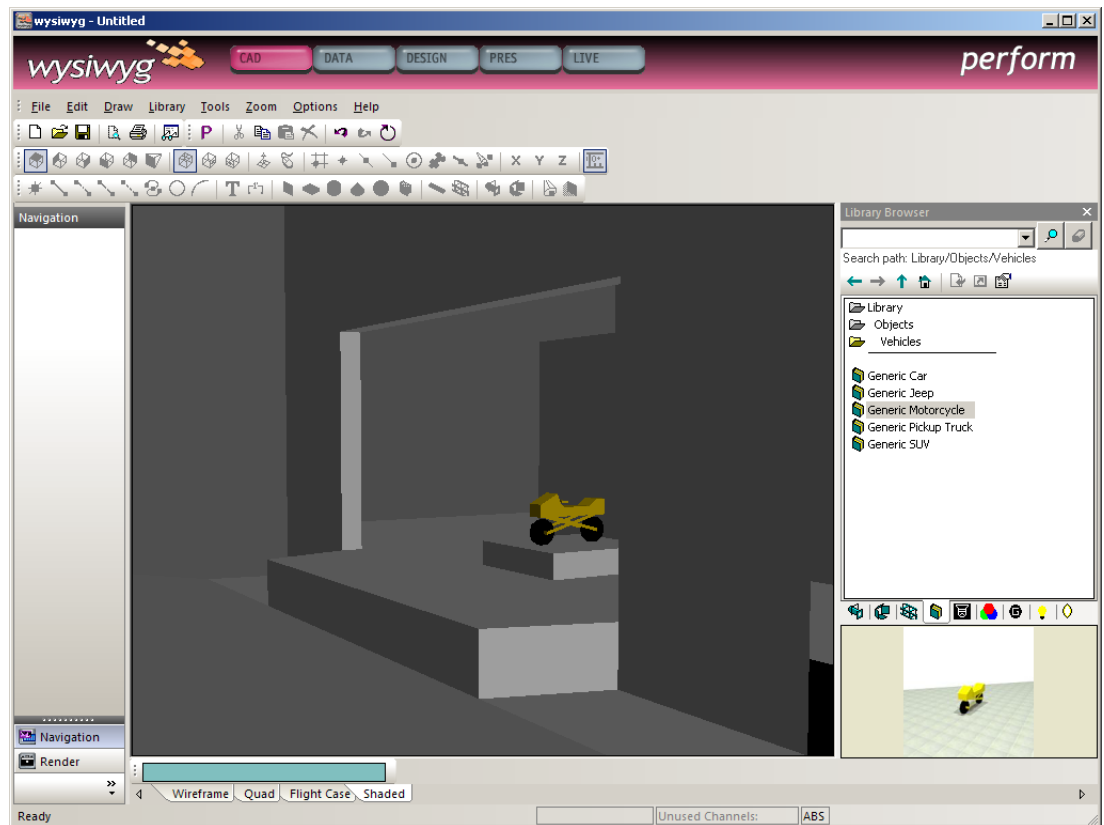
- 3 Specify the coordinates for the end point of the line (for example, 20,20,0), and then press ENTER.
- 4 Right-click and select **Finish Line**.

To extrude a line into a surface

- 1 With the line still selected, from the **Tools** menu, click **Extrude**, and then choose **Lines into surfaces**.
- 2 Enter the value for the extrusion distance, for example 20, and then click **OK**.

Result: The line is stretched into a surface.

Switch to Shaded view to view your final set design. Your set should look similar to the following:





Lesson 3 - Defining hanging positions

Introduction

Once the set is in place, you can define your hang structures. All fixtures in WYSIWYG must be placed on a hang structure, which can be either a pipe or truss.

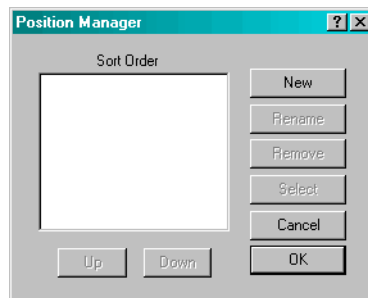
In this lesson you will learn how to insert a hang structure and define the position names in your document.

Step 1 - Defining a position name

Position names are used to identify hang structures. To organize and sort the position names, use the Position Manager available in WYSIWYG. The order in which the position names appear in the Position Manager defines how they are sorted in reports.

To define a new position name

- 1 From the **Edit** menu, choose **Positions**.



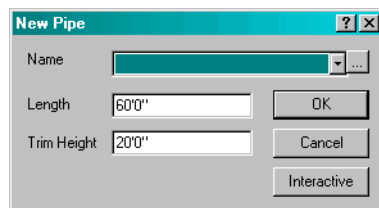
- 2 Click *New*, and then enter the name of the position.
- 3 Repeat step 2 for all position names as desired.
- 4 To sort the position names, use the *Up* and *Down* buttons to move the selected name in the list.

Step 2 - Drawing a pipe

Pipe hang structures can be drawn using the **Draw** menu.

To draw a pipe

- 1 From the **Draw** menu, choose **Pipe**.





- 2 Choose the associated position name as defined in "To define a new position name" on page 17.

- 3 Specify the dimensions for the pipe (for this example, accept the defaults), and then click *OK*.
- 4 Click or type to place the pipe in your drawing.

Step 3 - Inserting truss

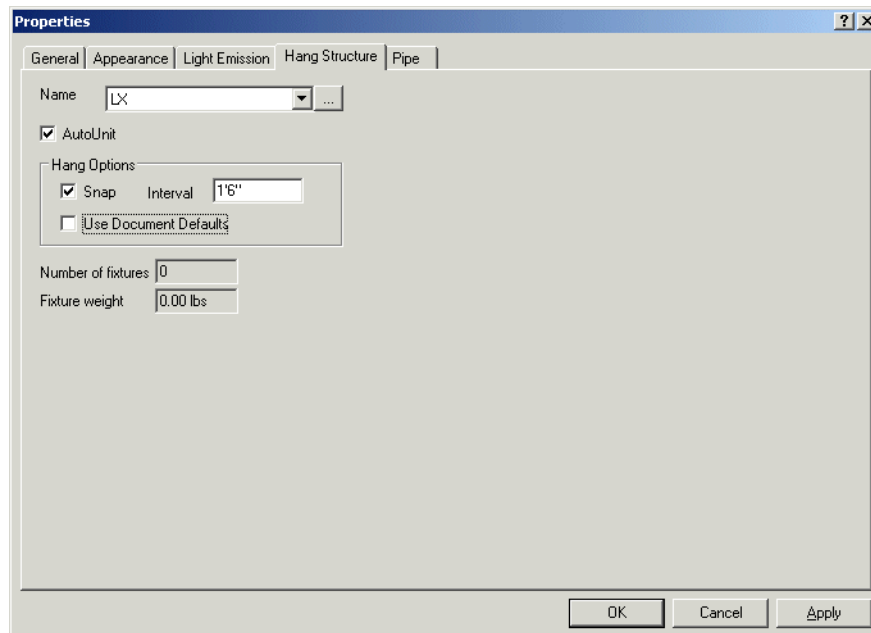
Truss hang structures can be inserted from the library.

To insert truss

- 1 Activate the *Assembly snap* tool  to ensure truss pieces are bolted together.
- 2 From the **Library** menu, choose **Browse Library**.
- 3 Click the truss tool  at the bottom of the Library browser to ensure that you are browsing the truss library
- 4 Navigate to the desired piece, and then double-click in your drawing to insert it.
- 5 Hover your cursor over your drawing where you want to insert the object. Press TAB to set the missing coordinate before clicking to place the piece of truss in your drawing.
- 6 Click to place the first piece of truss.
- 7 To assemble subsequent pieces, hover the cursor over the end of the piece of truss you want to attach it to and it will automatically snap into position.
Tip: It may be useful to zoom in as needed.
- 8 Once the piece has snapped into position, click to insert it into the drawing.
- 9 When you are finished placing truss, right click and select **Finish Placing Truss**.

To assign a position name

- 1 Double-click the truss structure to open its properties.



- 2 On the *Hang Structure* tab, from the Name drop-down box, select the desired name.
- 3 Click *OK*.



Lesson 4 - Hanging and focusing fixtures

Introduction

After inserting hang structures into your drawing, you are now ready to hang and focus fixtures. A library of fixtures is available for you to choose from.


In this lesson you will learn how to:

- hang fixtures
- focus fixtures

Step 1 - Hanging fixtures

Fixtures are objects on the plot and must be hung from a hang structure.

To hang fixtures

- 1 From the **Library** menu, choose **Browse Library**.
- 2 Select the fixture tool at the bottom of the library browser to ensure that you are browsing the fixture library. 
- 3 Navigate to the desired item, right-click, and then select **Insert**.
- 4 Move the mouse over a hanging structure (pipe or truss) and click to hang the fixture.
- 5 When you have finished placing the fixtures, right click and select **Finish Placing Fixtures**.
- 6 Repeat steps 3-5 for all desired fixture types.

Step 2 - Inserting a focus position

There are three methods for focusing fixtures:

- clicking and dragging the light beam of the selected fixture
- changing the pan and tilt values by double-clicking to access the fixture's properties box
- inserting focus positions

The method for inserting focus positions is described here.

Focus positions allow you to quickly attach a focus value to a fixture.

To insert a focus position

- 1 From the **Draw** menu, choose **Focus Position**.
- 2 Type a label for the new focus position and then click *OK*.
- 3 Press TAB to adjust the missing coordinate, if necessary.
- 4 Click to place the focus position in the drawing.

Step 3 - Focusing fixtures using focus positions

To associate a fixture with a focus position

- 1 From the **Tools** menu, choose **Quick Focus**.
- 2 Click on the focus position to set it as the active focus position, and then click on the desired fixture to set its focus position.
- 3 When you are finished focusing fixtures, right-click and select **Finish Quick Focus**.



Lesson 5 - Assigning fixture attributes

Introduction

In this lesson you will learn how to assign attributes to fixtures.

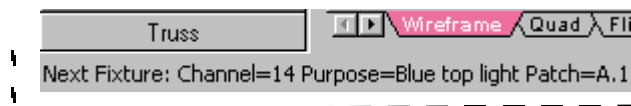
Step 1 - Assigning attributes to fixtures

Once you have fixtures in your drawing, you may want to assign data, accessories, or other attributes to them. To quickly assign values to fixtures, use Quick Tools, which is available in CAD mode.

To use Quick Tools

- 1 From the **Tools** menu, choose **Quick Tools**.

- 2 Select the tool(s) that you want to activate, and enter the initial value. When you have completed your selections, click *OK*.
Result: The cursor will show a "Q" to indicate that Quick Tools has been activated.
- 3 Click on the fixtures one by one to assign the specified information.
Tip: The prompt line at the bottom left corner displays the parameter information that will be assigned to the next fixture that is clicked on.



- 4 When you are finished using Quick Tools, right-click and select **Finish Quick Tools**.

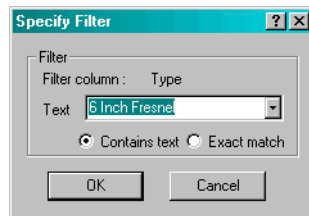
Step 2 - Filtering and modifying data

Any attribute associated with a fixture in your WYSIWYG drawing (for example, purposes, control channels, focus positions) is available for you to edit in Data mode. Any changes that you make are reflected throughout the entire file, including the drawing.

Begin in the spreadsheet. You can use data filters to quickly locate data in the spreadsheet.

To use a data filter

- 1 Right-click on the parameter heading for which you want to specify a filter (for example, the Type parameter).
- 2 Select **Add Data Filter**.



- 3 Select the value that you want to filter (for example, 6" Fresnel), and then click **OK**.
Result: The spreadsheet refreshes, displaying only the fixtures that meet the filter criteria.
- 4 Make any data changes that you require. To add or modify information, enter the data in the appropriate cell. You can add or modify chunks of information at the same time by selecting a series of cells and typing.
- 5 To remove the filter, right-click in the appropriate column, and then select **Remove Data Filter**.

Step 3 - Entering sequential numerical data

To enter sequential numerical data

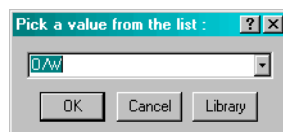
- 1 Enter the first number in the appropriate cell.
- 2 Press and hold the ALT key, and then press the down arrow to input subsequent values incrementally.

Step 4 - Choosing data

To choose data

- 1 If you are unsure of the exact value for parameters such as colour, gobo, and other parameters that make reference to library items, right-click the item, and then select **Pick value from the list**.

Result: This action opens a dialog with a pull-down list of the values currently used in your file.



- 2 Choose the value that you want, or click **Library** to make a new selection.

Note: This is only available for certain fields (for example, the Type column and Lens column).



Lesson 6 - Building lighting looks in Design mode

Introduction

In Design mode, you can create static lighting looks using the design tools. You can then save and render these looks to output photorealistic images. Note that you can only control fixture parameters that are ordinarily controlled by DMX.

You can also simulate cross-fading between lighting looks without the need of a lighting console by using the shortcut properties in the shortcut bar to specify the 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.

Once you create the look and specify the fade time, you can use the design tools to customize the look. 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.

In this lesson you will learn how to build a lighting look in Design mode and cross-fade to another lighting look.

Step 1 - Creating a new lighting look


To create a new look

- 1 In the Looks shortcut bar area, right-click and select **New Look**.
- 2 Type the name of the new look (for example, Scene1).
- 3 In the *Fade time* box, type the fade time in seconds for this look, and then click *OK*.
- 4 Create another look in the same way, this time with a different look name. Customize both looks in the following steps.

Step 2 - Using the Intensity tool

The zoom, intensity, and iris tools all work the same way.

To use the zoom, intensity, and iris tools

- 1 In your drawing, select the desired fixtures.
- 2 Open the Intensity tool, for example, by clicking the appropriate tool on the designer toolbar .




- 3 Click the dial, and then move the mouse up or down to set the intensity level, or click *Full* to set the intensity to 100%.

Step 3 - Using the Gobo tool

The gobo and prism toolbars work the same way.

To use the gobo and prism tools

- 1 Open the Gobo tool, for example, by clicking the appropriate tool on the designer toolbar .

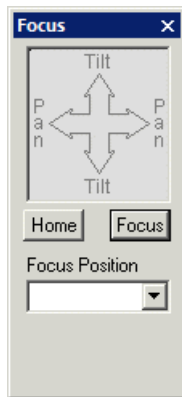


- 2 To assign a gobo, with the desired fixtures selected, select the desired wheel from the drop-down list, and then use gobo selection arrows to choose the desired gobo.

Step 4 - Using the Focus tool

To use the focus tool


- 1 Open the Focus tool by clicking the appropriate tool on the designer toolbar .

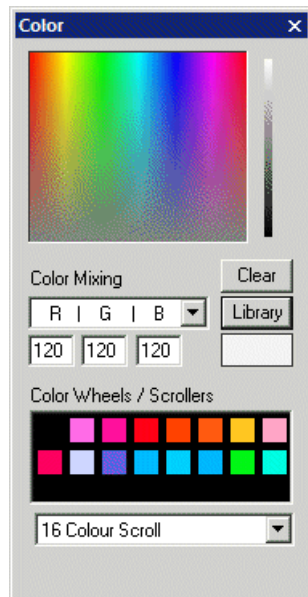


- 2 To focus automated fixtures, with the fixtures selected, choose from any of the following three methods:
 - Use the focus trackpad by clicking on the pad, and then moving the mouse.
 - Click the *Focus* button, and then click in the Wireframe drawing.
 - Choose a focus position from the drop-down list.

Step 5 - Using the Color tool

To use the color tool

- 1 Open the Color tool by clicking the appropriate tool on the designer toolbar .




- 2 To assign color, with the desired fixtures selected, click to pick a color on the palette or the color wheel.
- 3 If desired, you can use the RGB drop-down boxes to manually enter RGB, CMY, or HSI values.

Step 6 - Opening the Render Wizard

The Render Wizard steps you through the process of generating a rendering of your drawing.

To open the Render Wizard

- 1 With the desired lighting look active, open the Render Wizard by clicking on the appropriate tool on the designer toolbar .
- 2 In the right pane, click and drag the mouse to set up your image.
- 3 In the left pane proceed through the Wizard steps by using the *Next* and *Back* buttons.

For more information on the options available for rendering, consult the *WYSIWYG Reference Guide*.

Step 7- Cross-fading between lighting looks

After setting up at least two lighting looks, you can “fade” between them by using the cross-fade feature.

- 1 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**.

- 2 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 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**.



Lesson 7 - Creating and modifying reports

Introduction

In Presentation (Pres) mode, a series of pre-formatted reports is 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 in these reports. You can, however, modify the setup and layout of these reports.

In this lesson you will learn how to work with reports.

Step 1 - Entering show information

Shortcuts to the pre-formatted reports in Presentation mode are available in the shortcut bar. Click the desired report shortcut to open it. Show information such as venue, designer, show, and assistant is displayed at the top of each report.

To add or modify show info

- 1 From the **Options** menu, choose **Show Options**.
- 2 Click the *Show Info* tab.

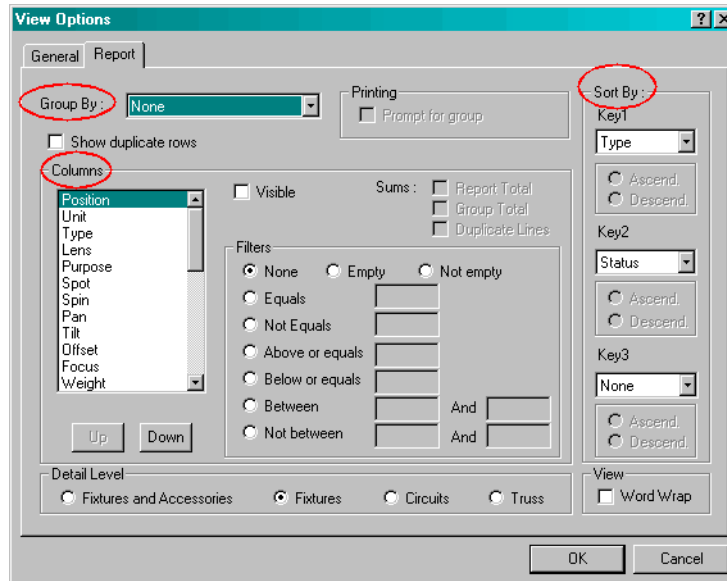
The screenshot shows a dialog box titled "Show Options" with a standard Windows-style title bar (help, close). The dialog is divided into several tabs: "Object Settings", "Report", "Data Error", "Regional Settings", "Document Summary", "Show Info", and "Object Defaults". The "Show Info" tab is currently selected and active. This tab contains four text input fields, each with a label to its left: "Venue", "Show", "Designer", and "Assistant". All input fields are currently empty. At the bottom of the dialog, there are two buttons: "OK" and "Cancel".

- 3 Make the necessary changes or additions (for example, add John Smith as the lighting designer).
- 4 Click *OK*.

Step 2 - Modifying a report

To modify a report

- 1 In the shortcut bar in Presentation mode, click a shortcut to open a report.
- 2 Right-click in the open report, and then choose **View Options**.
Result: The *View Options* dialog opens with the *General* and *Report* tabs available.
- 3 Click the *Report* tab.



Group By

The *Group By* drop-down list enables you to select a data field for grouping the data. These groups appear as separate tabs at the bottom of a report. Choose *None* to have all information displayed on the same page.

Columns

The *Columns* list displays the columns available for inclusion or exclusion in a report. You can display the fields in any order by arranging the order of this list using the *Up* and *Down* buttons. The top of the column list represents the left column on the page and the bottom of the list is the right column. To move a column up or down the list, click the column name and then use the *Up/Down* buttons.

Sort By

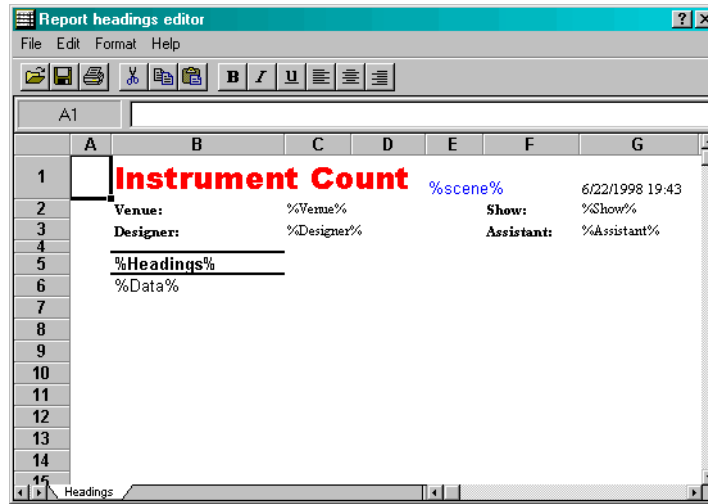
The *Sort By* list contains the settings for how the report is sorted. The keys that you specify are sorted in alphabetical or numerical order. When fixtures have the same value in the first sort key, the report is then sorted by the second sort key, and so on for the third sort key.

When you have set all the options, click *OK* to save and exit. The report refreshes and any changes are reflected immediately.

Step 3 - Formatting report headings

To format the report headings

- 1 Right-click in the report, and then choose **Redesign headings**.



- 2 Select the text to be formatted.
- 3 From the **Format** menu, choose alignment, font, border and color controls to customize the heading text. For example, change the font and color for the title "Instrument Count".
- 4 From the **File** menu, choose **Save**.
- 5 Close the *Redesign Headings* dialog box.

Result: A message box ask if you want to apply the heading to all reports in the document.
- 6 Click *Yes* to indicate that all reports will be modified. Click *No* to change only the chosen report.
- 7 Adjust column widths by clicking and dragging the column borders.
- 8 From the **File** menu, choose **Save**.



Lesson 8 - Working in the New Plots view

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.

Note: You cannot change the properties of items that you have inserted in CAD mode; you can only change the properties of items that you have inserted in the New Plots view.

In this lesson you will learn how to work with New Plots.

Step 1 - Creating a New Plot

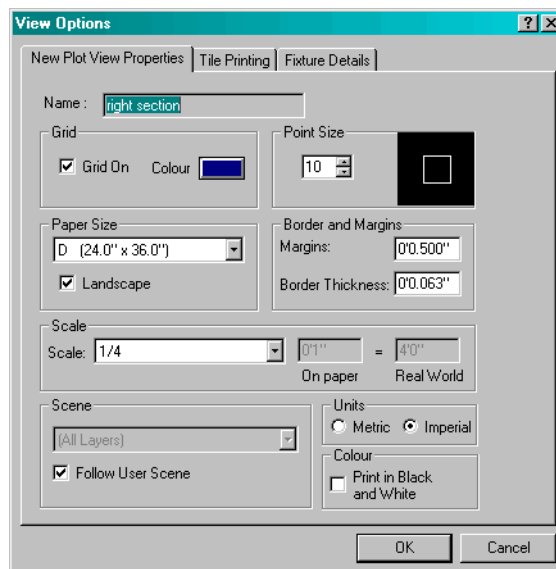
To create a New Plot

- 1 In Presentation mode, click the *New Plots* shortcut tab.
- 2 Right-click in the *New Plots* shortcut area, and then choose the type of plot that you want to create.
- 3 Type a name, and then click *OK*.

Note: By default, empty layouts are set up to use 24” x 36” sheets of paper.

To modify the paper size

- 1 In your New Plot, right-click in the report area, and then choose **View Options**.



- 2 From the *Paper Size* drop-down list, select the desired paper size.
- 3 Click *OK* to save and exit.

Next, you can determine the parts of the drawing that you want to see on the page by moving the page.

To move the page

- 1 Right-click in the New Plots work area, and then choose **Move Page**.
Result: The cursor turns into a hand symbol.
- 2 Hold down the left mouse button to grip the page and drag to move it until the parts of the drawing that you want are contained within the grid area.
- 3 When you are finished, right-click and choose **Finish Move Page**.

Step 2 - Manipulating objects on the plot

A New Plot is a snapshot projection of your CAD drawing. Once the snapshot is taken, you can modify objects on the New Plot without affecting the original CAD drawing.

Tip: Zoom in by pressing PAGE UP or by using the middle mouse button to facilitate the set-up.

To manipulate the objects on a plot

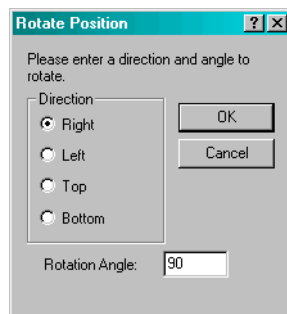
- 1 In the New Plot, select an object.
- 2 Click and drag to modify its position; to modify the layer properties, right-click the object, and then choose **Object Properties**.
- 3 Modify the properties as required.
- 4 Click *OK*.

Step 3 - Plotting non-horizontal hang structures

In the New Plots view, you can rotate non-horizontal hang structures, such as booms and ladders, around their base to show all mounted fixtures. This rotation can be right, left, top, or bottom.

To plot non-horizontal hang structures

- 1 Select the hang structure.
- 2 Right-click and select **Rotate Position**.



- 3 Select the direction and angle in which you want to rotate the hang structure.
- 4 Click *OK*.

Step 4 - Inserting objects

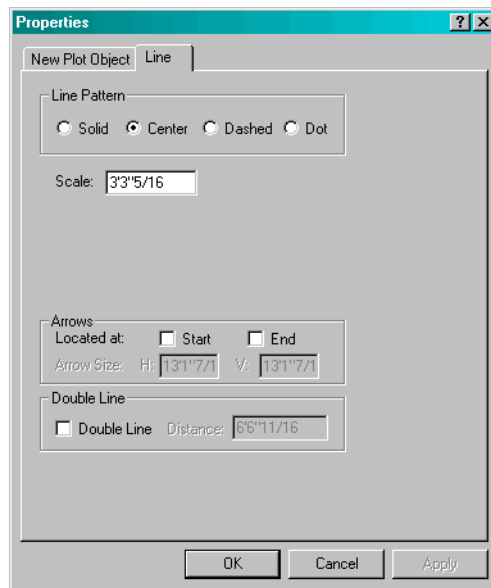
You can add objects to the plot for notation by simply drawing or selecting them from the **Draw** menu. Objects that you can draw include lines, arcs, circles, and text labels. You can also insert symbols from the Library Browser.

To insert objects

- 1 From the **Draw** menu, choose the object that you want to insert.
Note: Choosing **Symbol** opens the Library Browser.
- 2 In the properties box that opens, type the properties of your object.
Note: If you choose Interactive, you can change the size of the object manually.
- 3 Click *OK*, and then click in the plot to place the object.
Tip: You can resize objects by clicking and dragging them.

To add arrows to lines

- 1 Select the line to which you want to add one or more arrows.
- 2 Right-click and select **Object Properties**.
- 3 Click the *Line* tab.



- 4 Select where you want the arrows to appear, either at the start or end of the line, or both places.
- 5 Click *OK*.

To change a single line to a double line

- 1 Select the line that you want to modify.
- 2 Right-click it, and then select **Object Properties**.
- 3 Click the *Line* tab.
- 4 Select the *Double Line* check box.
- 5 Click *OK*.



Lesson 9 - Creating and modifying layouts

Introduction

In WYSIWYG, layouts are created by arranging various CAD, report, image, and other items on a defined paper size.

In this lesson you will learn how to work with layouts.

Step 1 - Creating a new layout

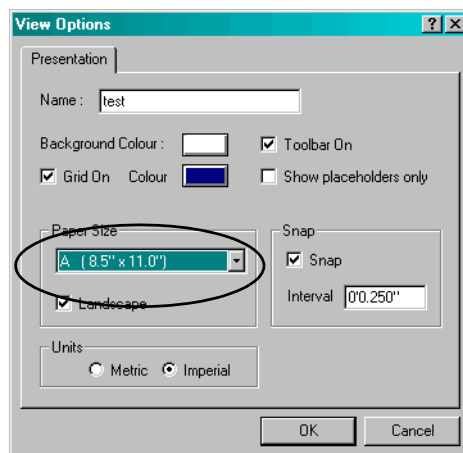
To create a new layout

- 1 In Presentation mode, click the *Layouts* shortcut tab.
- 2 Right-click in the *Layouts* shortcut area, and then choose **New Layout**.
- 3 Type a name, and then click *OK*.

Note: By default, empty layouts are set up to use 8.5" x 11" sheets of paper.

To modify the paper size

- 1 Right-click in the layout area and choose **View Options**.



- 2 From the *Paper Size* drop-down list, select the desired paper size.
- 3 Click *OK* to save your changes and exit.

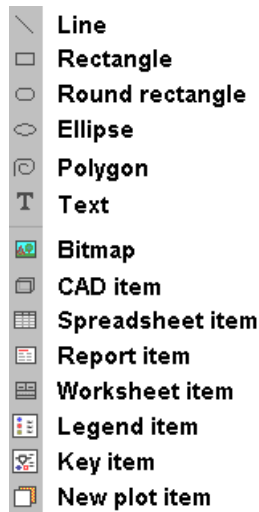
You may now begin placing objects on the sheet of paper. Use the *Pres Insert* toolbar to facilitate the process.

Tip: Zoom in by pressing PAGE UP to facilitate the setup.

Step 2 - Inserting objects into the layout

To insert objects on the layout

- 1 To insert an object, select the appropriate icon from the *Pres Insert* toolbar.



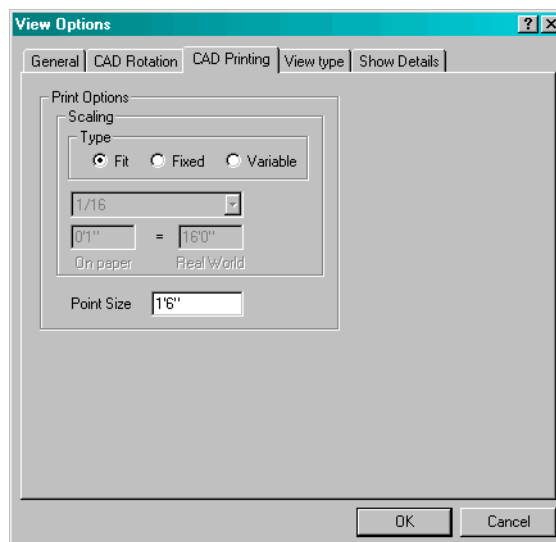
- 2 Click and drag a window (placeholder) on the layout where you want the item to appear.
- 3 Repeat this procedure for all desired items.

Step 3 - Modifying CAD items

To modify CAD items

In this example you will modify the scale of a drawing item.

- 1 Select the CAD item, right-click, and then choose **Content Properties**.
- 2 Click the *CAD Printing* tab.



- 3 Click the *Fixed* option button.
- 4 From the drop-down list, select *Custom*.
- 5 Type the scaling to be used when printing the drawing, for example, fixed at $1/8'' = 1'$.

- 6 Select any other desired items, and then click *OK*.

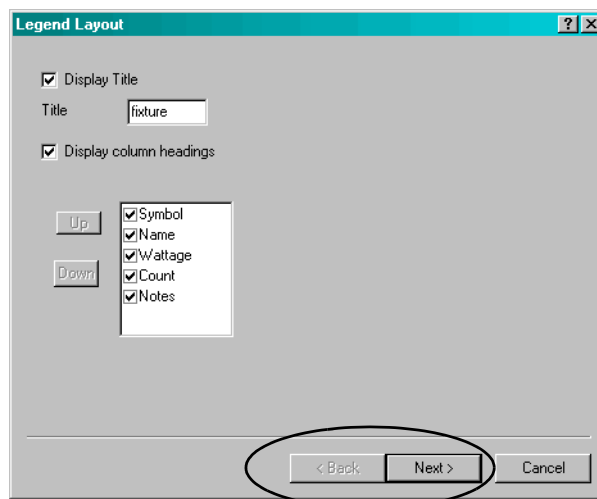
Step 4 - Adding legends and keys

The Legend and Key Wizards guide you through the process of adding a legend or key to your layout.

- **Legends** You may want to add a legend to your layout to provide a definition of the symbols used in the layout. The legend you create can include the symbol name, wattage, count, and any other special notes.
- **Keys** You may want to add a key to your layout to provide a definition for the fixture notation used in the layout. The key you create can include the symbol of one given fixture and identifiers for the various attributes notated around the symbol.

To open the Wizards

- 1 In the shortcut pane, click Wizards.
- 2 Right-click in the Wizards shortcut bar, and then select the appropriate item.
- 3 Type a new name for your legend or key, and then click *OK*.
- 4 Proceed through the steps using the *Next* and *Back* buttons.



To place the legend or key

- 1 On the *Insert* toolbar, either the Legend or Key icon.
- 2 Click and drag a window placeholder on the layout where you want the legend or key to appear.

Result: A message box prompts you to select the legend or key from the list.
- 3 Highlight the legend or key, and then click *Select*.



Lesson 10 - Patching

Introduction

In WYSIWYG, you can create two types of patch universes:

- **DMX** When you create this type of patch universe, you can patch fixtures or movement axes to it by assigning them a DMX address. The patch screen in Data mode is a visual representation of your DMX patch setup and is also used to monitor DMX activity while connected to a console.
- **Motion Control** When you create this type of patch universe, you can patch movement axes to it, and then use a motion control system (or WYSIWYG's Motion Control Console) to control the movement of any objects that are attached to the axis.

In this lesson you will learn how to patch fixtures and read a resulting patch universe.

Step 1 - Creating a new patch universe

To create a new patch universe

- 1 In Data mode, click the Patch tab.
- 2 Click the *Patch* shortcut bar.
Result: The Patch shortcuts are displayed.
- 3 Right-click in the shortcut bar, and then select **New Patch**.
- 4 Type the name of the patch universe. Patch universes can be named anything. For example, "A", "Dimmers", "Dim", "Movers", "Scrollers", and so on.
- 5 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.
- 6 Click *OK*.

Result: The patch shortcut appears in the Patch tab.

Step 2 - Patching fixtures and movement axes in the patch layout screen

To patch fixtures and movement axes in the patch layout screen

- 1 In the CAD drawing in the top portion of the window, click to select the desired fixture or movement axis.
- 2 Click and drag the fixture/movement axis into the DMX or motion control patch universe at the bottom of the window.
- 3 Repeat for all fixtures/movement axes as desired.

Step 3 - Repatching a fixture

To repatch a fixture

Select the desired fixture by clicking its first DMX attribute directly in the patch universe, and then drag it to a new address.

Step 4 - Reading the patch

Reading the DMX patch

The name of the patch universe is displayed in the title bar.

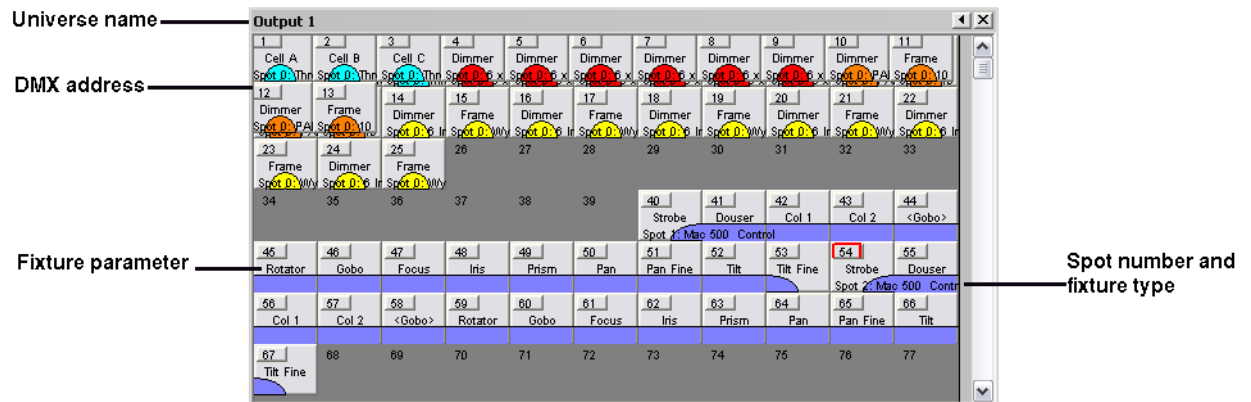
Each box in the patch universe represents one DMX channel. Fixtures are represented by colored bumps that span the number of required DMX channels.

If you are connected to a console and are receiving a DMX signal, the incoming DMX values are displayed in the top right corner of each box.

The number in the upper left corner of each box is the DMX address.

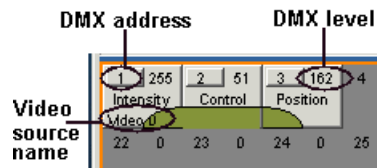
The fixture parameter controlled by each channel is identified in the center of the box.

The spot number and fixture type span the center of the box.



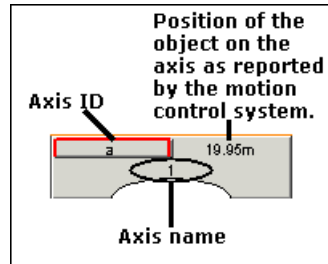
Reading the patch for streaming video sources

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 movement axis, the patch appears as shown below:



- For linear movement 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.



Lesson 11 - Inserting and connecting to a console

Introduction

WYSIWYG enables you to visualize and pre-cue your lighting show in real time. To use Live mode, you must be connected to a DMX console, compatible off-line editor, or a motion control system.

Note: Ensure that your hardware and/or software is properly set up.

In this lesson you will learn how to insert a console and connect that console in Live mode.

Step 1 - Inserting a console

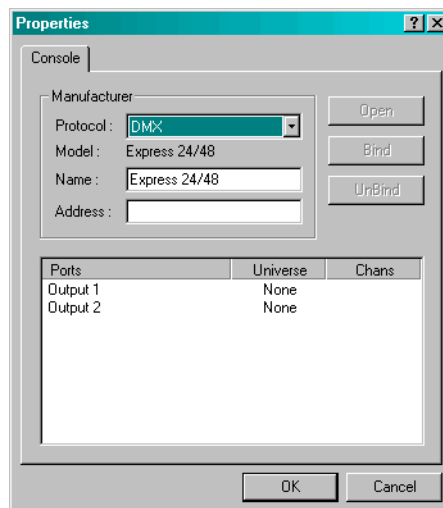
To insert a console

- 1 From the **Live** menu, choose **Device Manager**.
- 2 Click *New*.
- 3 Expand the Manufacturer section, and then choose the desired console (for example, Express 24/48).
- 4 Click *Insert*.

Step 2 - Connecting to a console

To bind the console output to the patch universe

- 1 In the Device Manager window, highlight the console, and then click *Properties*.



- 2 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.

- 3 In the *Address* box, type the designated address of the console, if applicable.

- 4 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."
- 5 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.
- 6 Select the desired Patch Universe, and then click *Select*.
- 7 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.
- 8 Click *OK* to close the *Properties* box.
- 9 Click *Connect*.
Result: The status of the console changes to "connected."
- 10 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.



Lesson 12 - Using streaming video

Introduction

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.

In this lesson you will learn how to

- configure a new video source with the Video Manager
- draw the screen on which the video will play and assign the video source to it
- use the Video design tool to play the video
- use a console to play the video

Step 1 - Configuring a new video source

To configure a new video source for streaming video

In this step you create a new video source for streaming video with the Video Manager. You can select a video file or you can capture a live video stream from an external source, such as a video capture device that is attached to your computer (provided that you have WYSIWYG Perform).

For video files, you must select one of the following video file formats for playback:

- Motion Picture Experts Group (MPEG)
- Audio-Video Interleaved (AVI)

Note: If you cannot view these file types, you may not have the proper decoder installed on your PC. Install the appropriate decoder, and then try viewing the file again. For help on installing the decoder, consult the developer of the video file type.

For live video streams, your capture device must be using WDM drivers.

Note: You can only incorporate live video if you have WYSIWYG Perform; you cannot select a video from a live capture device if you have WYSIWYG Design.

1 Click **Tools** > **Video Manager**.

2 Click *New*.

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 installed on your computer, click *Video Capture*, and then use the drop-down arrow to select the appropriate device. (This option is available only if you have WYSIWYG Perform installed.)

Note: The video capture device must already be configured on your computer and must be using WDM drivers.

- 4 To play the video with a DMX control, 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: You must have a named patch already configured on your computer.
- 5 Click *OK*.
- 6 In the Video Manager, ensure that the status of the video source is *Connected*. If it is not, then press *Connect*.
- 7 Click *OK* to save your changes and close the Video Manager.

Step 2 - Drawing a screen and attaching the video source

To draw a screen and attach the video source

- 1 From the **Draw** menu, click **Screen**.
- 2 In the appropriate boxes, type the width and height of the screen (or accept the default width of 8 feet and height of 6 feet).
- 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, click the *Image File* option button, and then click *Browse* to locate the graphic.
 - To attach the video source that you have just configured to the screen, click the *Video Source* option button, and then choose the video source from the drop-down box.
- 4 Click *OK*.

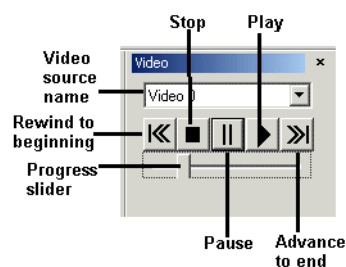
Step 3 - Using the Video design tool to play the video

After you draw the screen and attach the video source, 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 design tool to play the video

Once you have the video source attached to the screen, you must use the Video design tool to control it.

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 design 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 within Live Mode before using any of the Video design tool commands.



- 1 Click the Design mode button.
- 2 Click the Shaded tab to view the video screen(s) that you have drawn.
- 3 Click the Video design tool icon to open the tool window.
- 4 From the drop-down list, select the video source.
- 5 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 press the *Pause* button, the frame of the video that is playing at the time is held on the video screen; when you press the *Stop* button, the video stops playing and the screen goes blank and the video returns to the beginning.

Step 4 - Controlling the video with a console device

You can also patch the *control* of the video source to a console device. 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.

Note: To control the video source with a console device, you must have selected *DMX Patch* in the Video Source window in Step 1, and then chosen the patch universe for the video source. Also, your console device must be connected.

To control a DMX patched video source with a console

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.

To control the video source, the DMX patch is allocated three channels, each given a different name in the patch window:

- The first channel is called *Intensity*.
- The second channel is called *Control*.
- The third channel is called *Position*.

For details on the channel levels, see the *Reference Guide*.

- 1 Click the *Live* mode button.
- 2 Click the *Shaded* tab.
- 3 Ensure that the console device is connected to the patched video source.
 - To play the video from start to finish, perform the following steps:
 - a. Set the first channel (Intensity) to a value between 128—255 (above 50%).
 - b. Set the second channel (Control) to a value between 170—255 (between 66% and 100%).
 - To pause the video, perform the following steps:
 - a. Set the first channel (Intensity) to a value between 128—255 (above 50%).
 - b. Set the second channel (Control) to a value between 170—255 (between 66% and 100%) and play the video to the spot where you want to pause it. Then move the second channel to a value between 85—169 (between 33% and 66%) to pause the video at this spot.
 - To control the progress of the video manually, perform the following steps:
 - a. Set the first channel (Intensity) to a value between 128—255 (above 50%).

- b. Set the second channel (Control) to a value between 1—84 (between 1% and 33%).
- c. Slide the third channel (Position) up and down to control the progress of the video, forward and backward.

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 device from Live Mode before using any of the Video Designer Tool commands.



Lesson 13 - Using moving scenery

Introduction

You can view moving scenery in WYSIWYG by creating linear or rotation movement axes, attaching objects to them, patching them in Data mode, and then using a motion control system (or WYSIWYG's Motion Control Console) to define the object's position along the axis. You can attach objects such as risers, library objects, or custom objects that you have drawn.

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 movement axes; however, you can attach pipes that hold fixtures to axes.
- Currently, you cannot attach focus positions or groups containing focus positions to movement axes. However, in future releases you will be able to attach either of these items to movement axes.

Step 1 - Drawing the movement axis

To draw a linear movement axis

- 1 In CAD mode 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 movement 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 CAD mode 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.

Step 2 - Attaching a movement axis to a patch universe

To attach a movement axis to a DMX patch universe

To move the object with a DMX control, you first have to attach the movement axis to a named DMX patch universe that you have created in Data mode.

Note: For information on creating a patch universe, see “Step 1 - Creating a new patch universe” on page 41.

- 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 Click the *DMX Patch* option button.
- 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 attach a movement axis to a motion patch universe

To move the object with a motion control system, you first have to attach the movement axis to a named motion control patch universe that you have created in Data mode.

Note: For information on creating a patch universe, see “Step 1 - Creating a new patch universe” on page 41.

- 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 Click the *Motion Patch* option button.
- 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*.

Step 3 - Attaching objects to the movement axis

To attach an object to a movement 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*.

Step 4 - Viewing moving scenery


You have two choices when viewing moving scenery:

- You can use the Moving Scenery designer tool and the Cross-Fade feature in Design mode to fade between looks containing moving scenery.
- You can use a DMX console or a motion control system in Live mode to control the movement of objects attached to patched movement axes.

To view moving scenery in Design mode

- 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 movement axis.
- 10 Click the slider to advance the object to the position where you want it to start on the selected axis.

Note: In the *Start* box, you can also 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 movement 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 movement 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 movement axis with a console in Live mode” below or “To control a motion-control patched movement axis with a motion control system in Live mode” on page 54.

To control a DMX patched movement axis with a console in Live mode

After you have patched a movement 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.
- 1** Ensure that the console device is connected to the patched movement axis. For details on connecting a console, see “Step 2 - Connecting to a console” on page 45.
 - 2** To move the object quickly along the movement 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.

To control a motion-control patched movement axis with a motion control system in Live mode

After you have patched a movement 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.

- 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 movement axis. You connect the Motion Control Console (and motion control systems) in much the same way as you connect consoles. For details, see "Step 2 - Connecting to a console" on page 45.
- 3 To change the settings of a patched movement axis, highlight the axis in the left pane, and then choose from the following settings:
 - **ID box:** To change the ID of the movement 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 movement 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 movement 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 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*.

