Quick Start Guide

Release 34 December 2014

Contents

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 14 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.

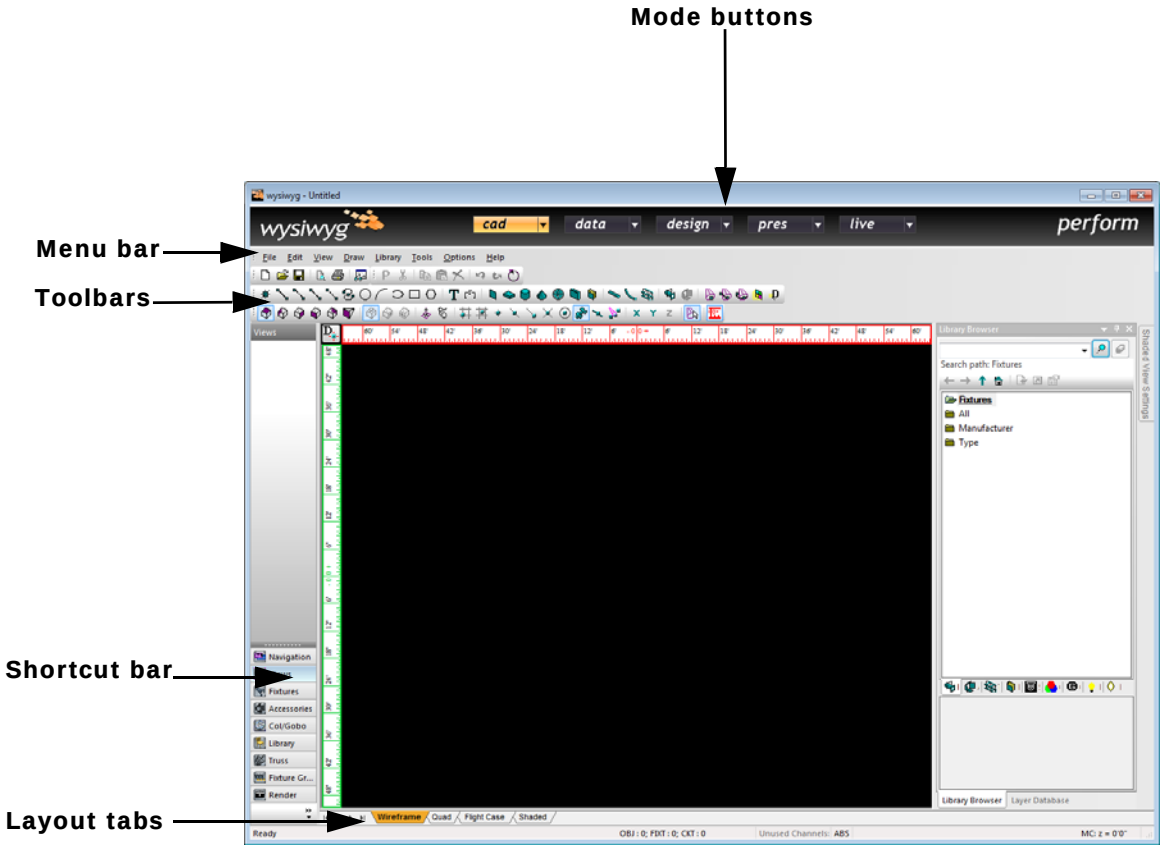
Step 1 - Starting to work in WYSIWYG

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 right and a list of files you have recently used are shown under Open File.

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:

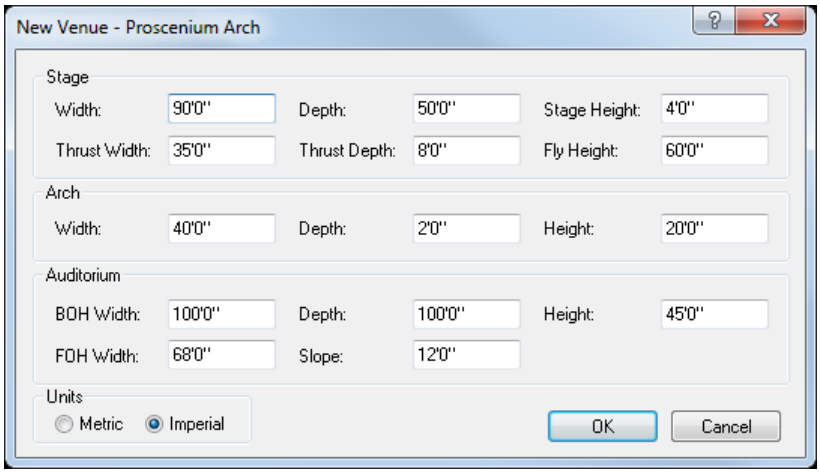
- insert a venue
- build your set using the following methods:
  - from CAD objects
  - from library items
  - using CAD tools

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.

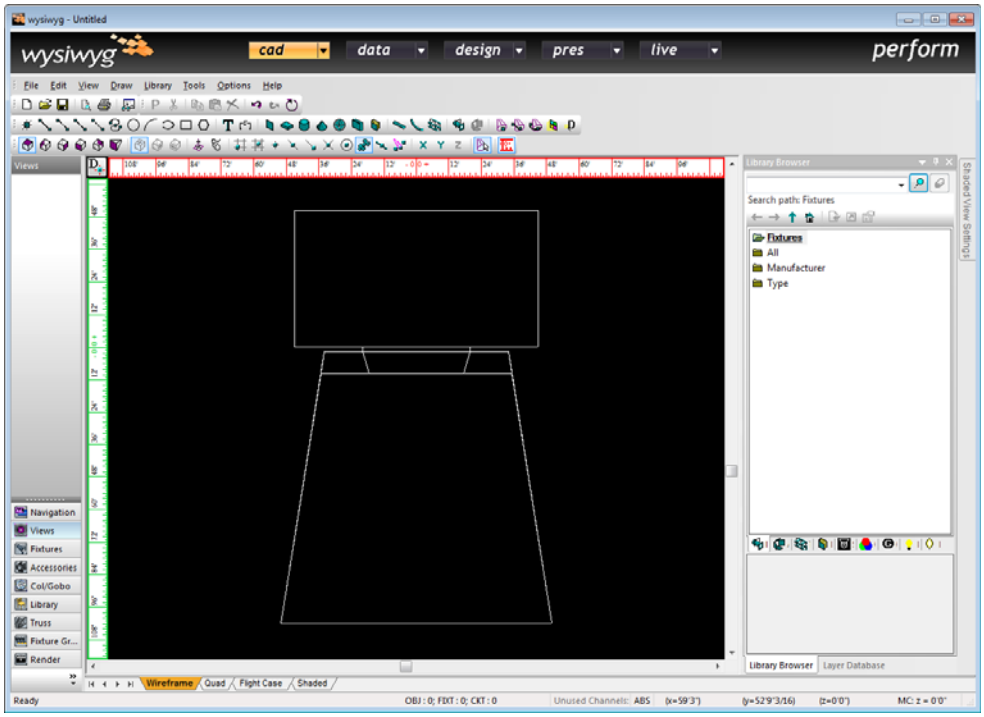
To insert a venue

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



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.*

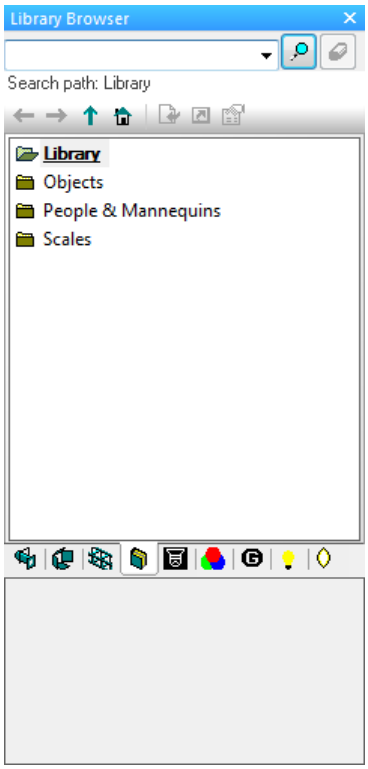
4 Click to drop the riser in your drawing.

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**.

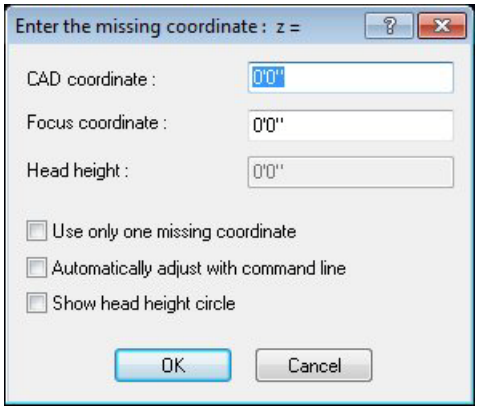


2 Select the *library item* tool  at the bottom of the library browser to ensure that you are browsing the scenic library.

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**.

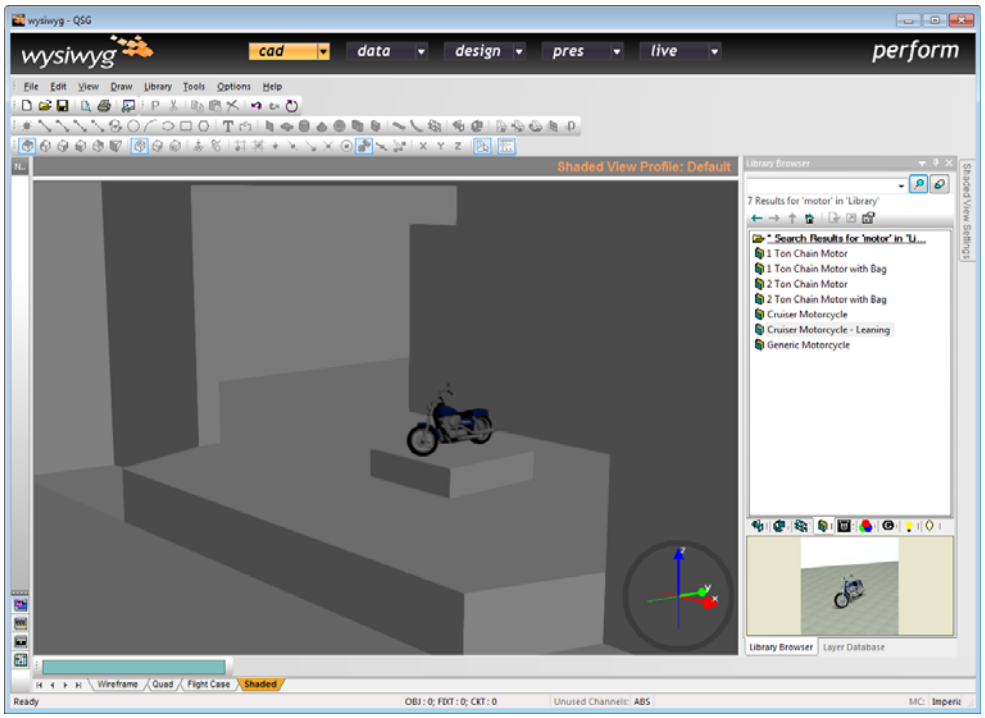
To extrude a line into a surface

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.

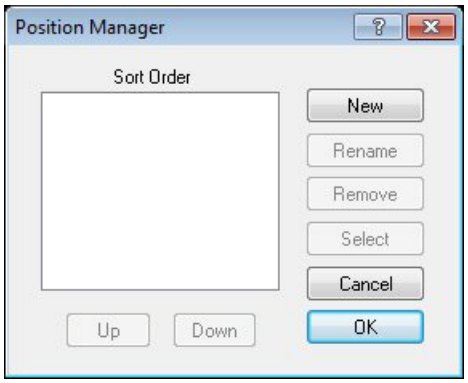
In this lesson you will learn how to insert a hang structure and define the position names in your document.

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.

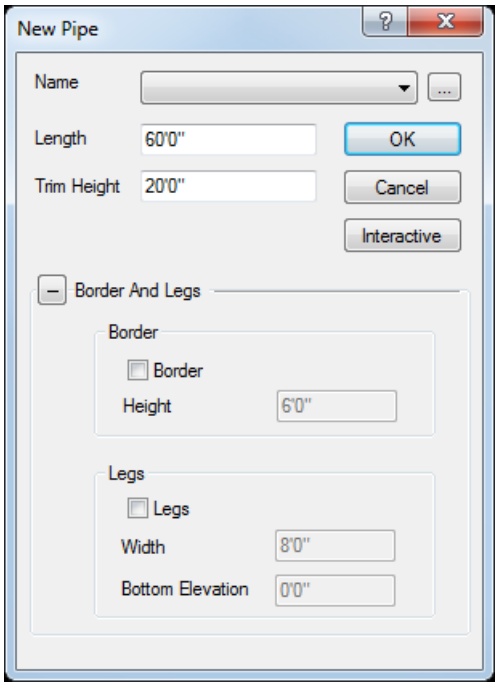
4 To sort the position names, use the *Up* and *Down* buttons to move the selected name in the list.

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 15.

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 is easiest to place your first truss on the left side of your screen, and continue to build your truss by adding new pieces to the right end.

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**.

Indicators for truss assembly

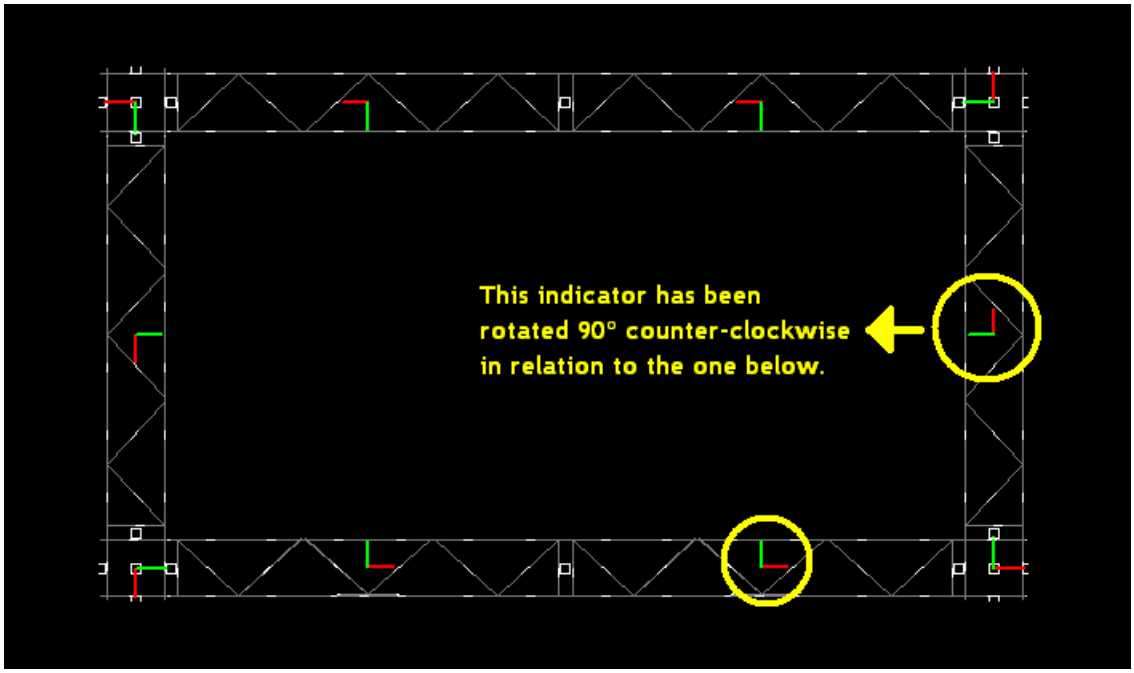
When hanging or selecting truss, a 3D indicator appears, displaying a directional/ positional reference for the individual piece of truss. You can use these indicators as a visual guide, helping you to assemble truss structures and instantly determine if a truss piece was not assembled as intended.

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.



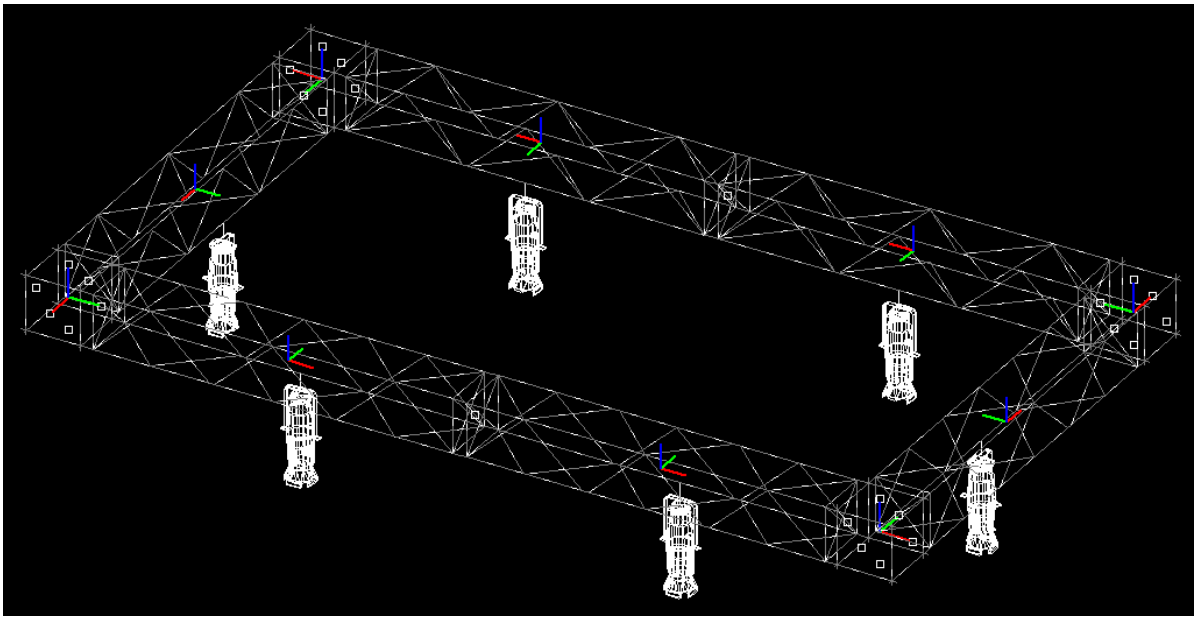
When the 90º corner block is used, the "running direction" of the truss changes. As such, the next truss piece is now rotated as shown by its indicator. This continues all the way around this rectangular truss structure until we connect back to the bottom left corner.

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 90º (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 90º counter-clockwise would cause the green line to point to your left and the red to point up, while rotating the indicator 90º clockwise would cause the green line to point right and the red arrow to point down.

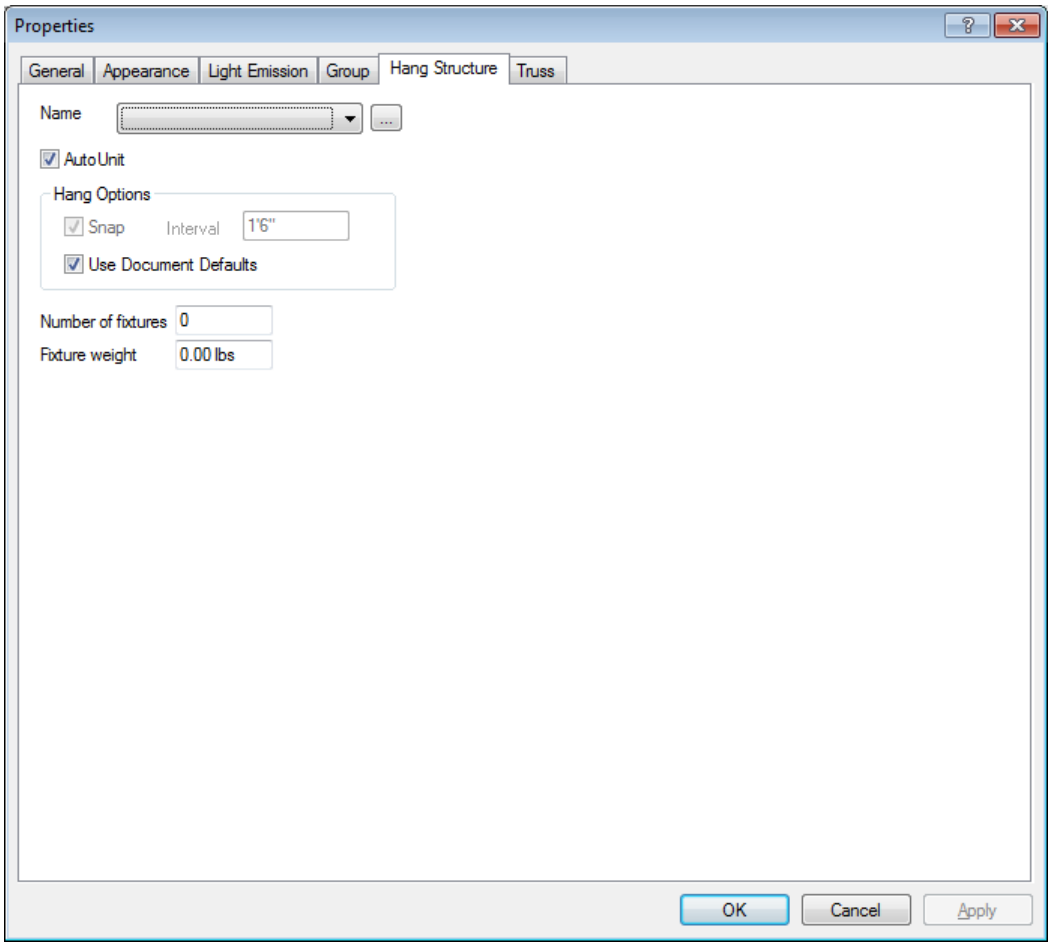
These are the only two cases in which fixtures hanging from the two pieces attached to the 90º corner block would behave “as expected”; should the indicators appear in any other positions, undo and assemble the truss again, making sure not to activate any of the right-click roll/alternate mountpoint 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.



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

Note: You can only focus conventional fixtures in CAD mode.

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.

The following is a brief description of each Quick Tool. For further information, you can also click the Help button (?) in the upper right corner of the Quick Fixture Tool window, and then click on the tool of your choice.

Descriptions of Quick Tools

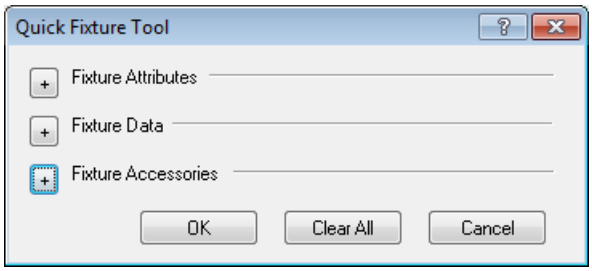
- Quick **Spot** enters a spot number in the Spot box.
- Quick **Unit** enters a unit number in the Unit box.
- Quick **Purpose** enters a fixture purpose in the Purpose box.
- Quick **Focus** enters a focus position or pan and tilt angle for the selected fixture.
- Quick **Channel** enters a channel number in the Channel box.
- Quick **Patch** enters a patch universe assignment in the Patch box.
- Quick **Dimmer** enters a dimmer number in the Dimmer box.
- Quick **Circuit** enters a circuit name and number in the Circuit Name and Num boxes.
- Quick **Color** enters a gel color number for the selected fixtures.
- Quick **Gobo** enters a gobo number for the selected fixtures.
- **Other** enters an accessory type for the selected fixtures.

Notes:

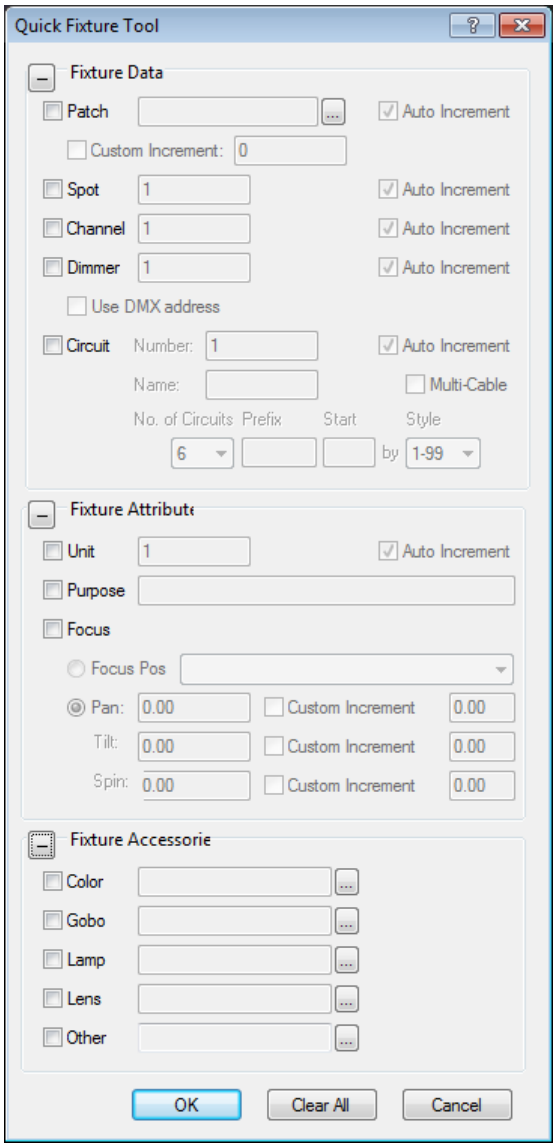
- When typing values, type the exact value or use the [...] browse button to open a selection dialog.
- The entry in the patch box must fulfill the requirements of patch notation which is universe.#.
- When using the quick tools to insert a color or gobo, WYSIWYG automatically inserts a default color frame or gobo holder, which is already defined in the fixture. Remove the default frame and add a new one using *Add Accessory*.
- 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.

To use Quick Tools

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



2 Click the plus signs (+) beside each heading to expand the section and work with the tools.

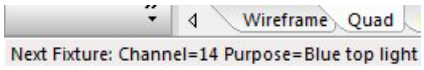


3 Select the tool(s) that you want to activate, and enter the initial value. When you have completed your selections, click *OK*.

Result: The cursor will show a “Q” to indicate that Quick Tools has been activated.

4 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.



5 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.

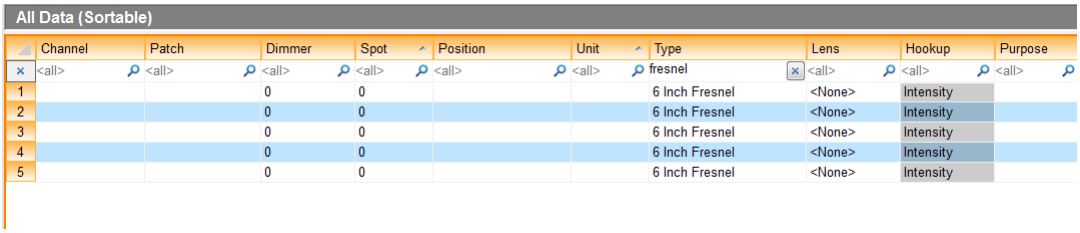
Begin in the spreadsheet. You can use data filters to quickly locate data in the spreadsheet.

To use a data filter

1 In the Spreadsheet, locate the Type column and specify a filter to locate the fixtures you want to work with.

2 In the Filter bar, enter the text "Fresnel" to filter the Spreadsheet.

Result: The spreadsheet refreshes, displaying only the fixtures that meet the filter criteria.



3 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.

4 To remove the filter, just click on the *X* button displayed on the right side of the Type column, beside the filter text you entered.

Step 3 - Entering sequential numerical data

To assign a sequential patch for a list of fixtures, 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 enter sequential numerical data

1 Select the fixtures you wish to patch sequentially by click and dragging their cells to select them.

2 Enter the starting patch in the following format, "**Universe Name.Starting Address +**".

Example: Type "**A.1 +**" and then press *ENTER*.

Step 4 - Choosing data

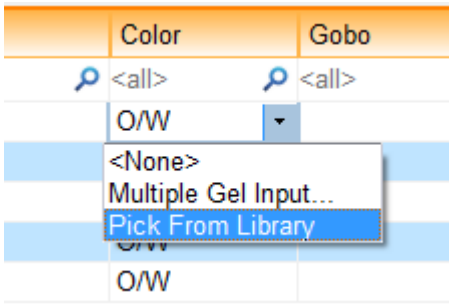
To choose data

1 In some columns a drop-down is available which lists relevant data.

Example: The Color column, a drop-down will appear displaying some options to help you enter the data into the cell(s).

2 If you wish to browse the library, select "*Pick from Library*".

Result: This action opens the *Color Select* window where you can browse the gel catalog to select your color.



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 .

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%. To save time, you can also use the following buttons to quickly adjust the intensity:

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 methods:

- Use the *Focus Pad* by clicking on the pad, and then moving the mouse.
- Use the *P / T Wheels* to control pan and tilt of the fixture.
- Click the *Focus* button, and then click in the Wireframe view at the point where you would like to focus the beam.
- Choose a focus position from the *Focus Objects* 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.

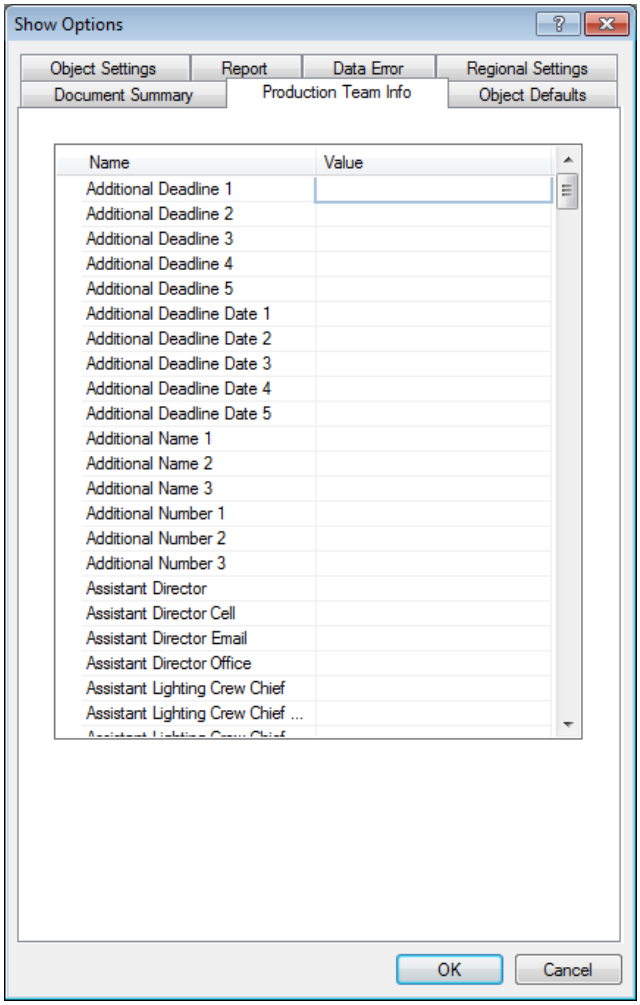
Step 1 - Entering show information

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 *Production Team Info* tab.



3 Click in the Value column and make the necessary changes or additions (for example, add John Smith as the lighting designer).

4 Click *OK*.

Step 2 - Modifying a report

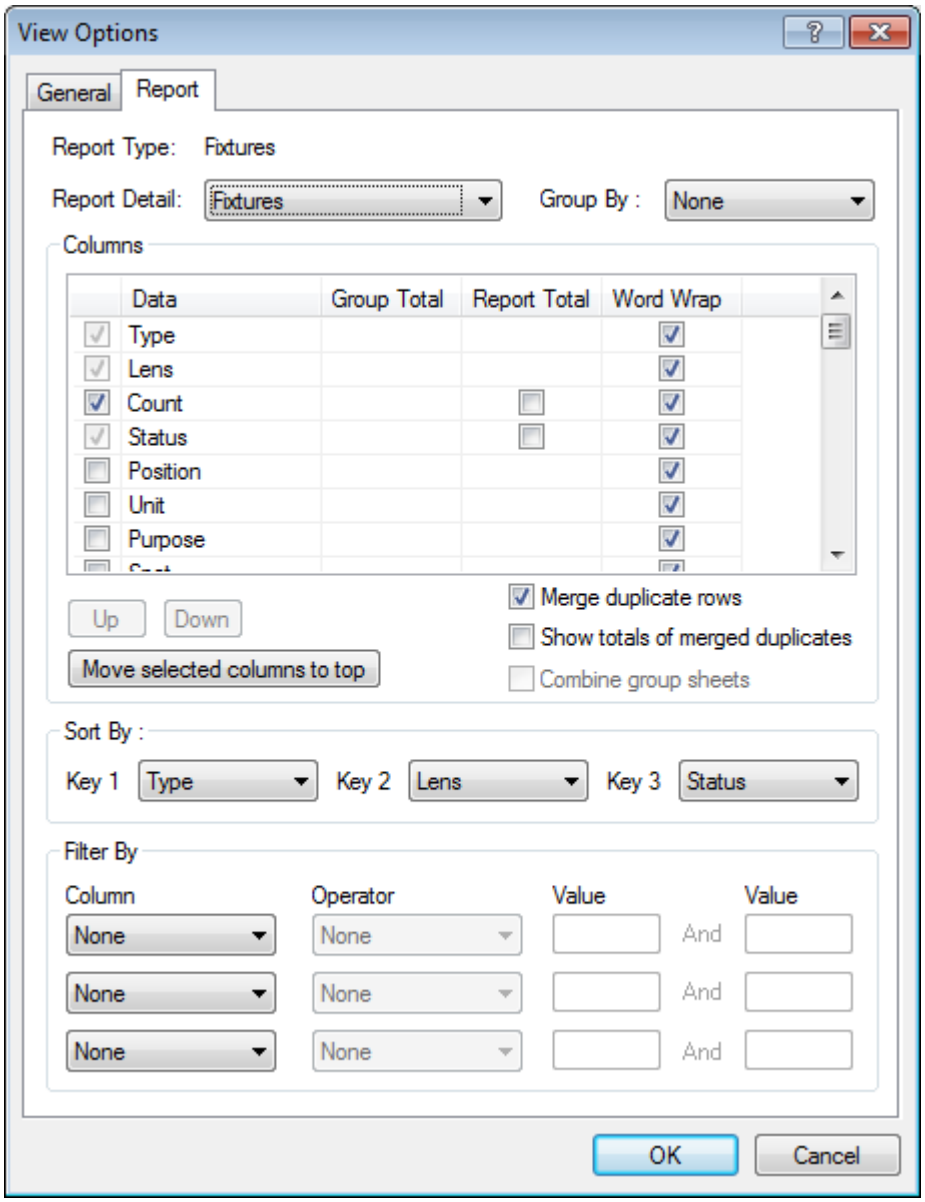
To modify a report

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

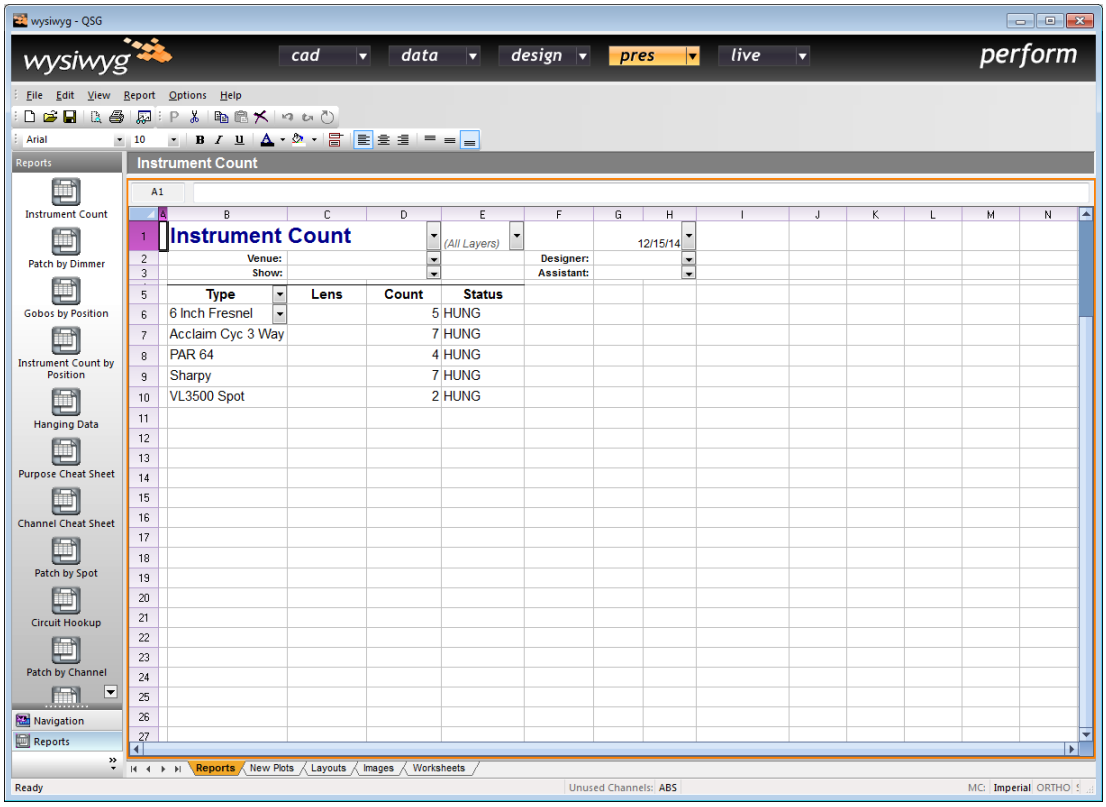
The *Columns* list displays the columns available for inclusion or exclusion in a report. You can display the fields in any order by arranging the order of this list using the *Up* and *Down* buttons. The top of the column list represents the left column on the page and the bottom of the list is the 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.

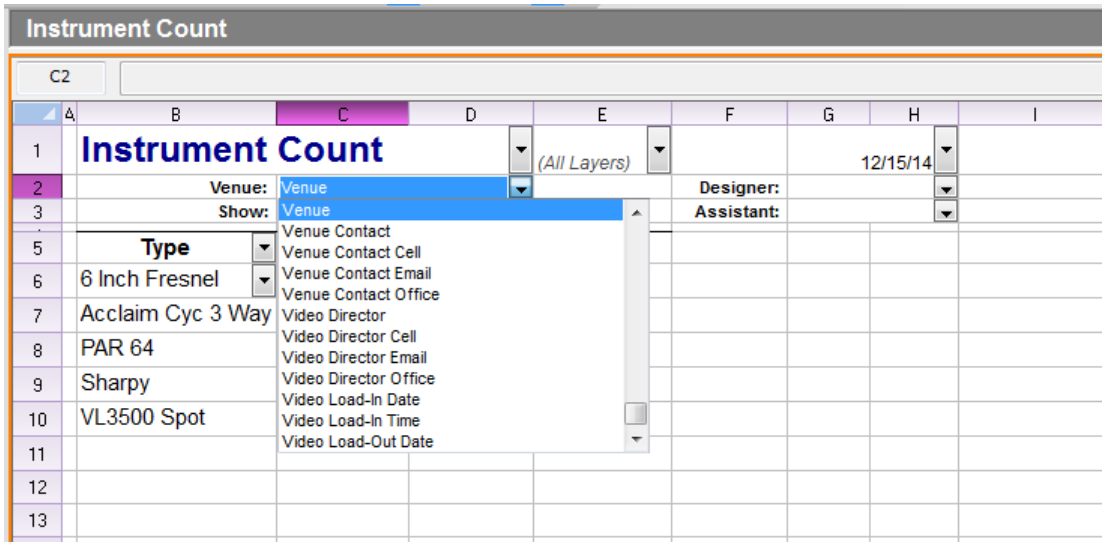
Step 3 - Formatting report headings



To format the report headings

1. In the report, select the text to be formatted.
2. From the Text toolbar, choose the desired alignment, font, text color, and cell fill options.
3. From the Report menu, select **Format Cells**, and then adjust the cell format, cell border, text alignment, font, and background colors as desired.

4 Select a Smart Cell drop-down and then choose the desired variable, as shown in the following graphic:



5 From the File menu, choose **Save**.

To format the report data

1 In the report, the cells below the column heading is the report data.

2 To format any of the columns, select a cell within a column, and then use the Text toolbar, to set desired alignment, font, text color, and cell fill options.

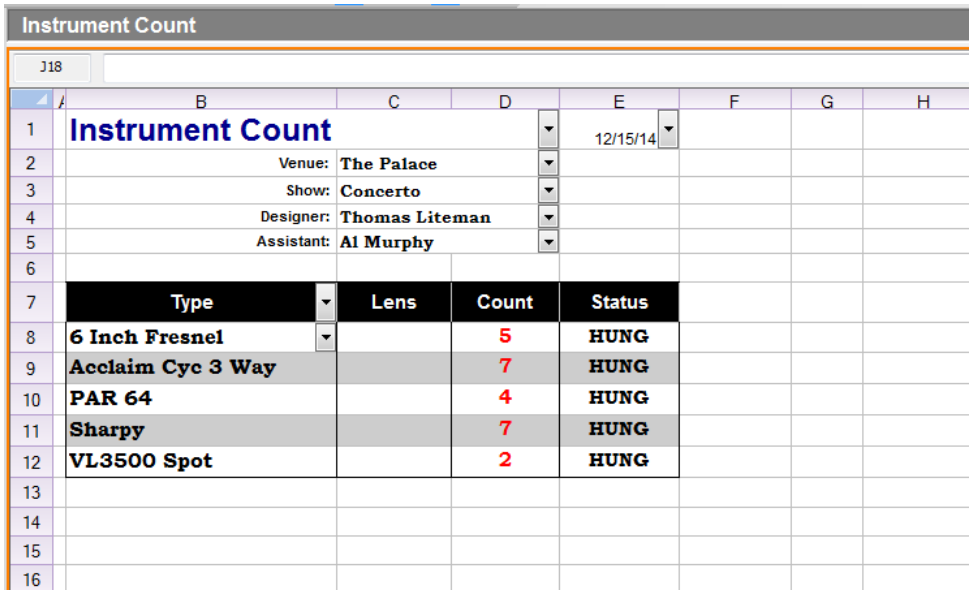
Alternatively, right-click on the data section and select Format Cells.

Note: To format cell borders, it is offered only from within the **Format Cells** dialog.

3 To turn on Zebra Striping, go to **Report** menu and select **Zebra Striping**.

4 Once you've completed your formatting changes, save your file.

Result: With some formatting to cells, and moving some of the data through copy/paste and row/column manipulation, you can customize the look of your Report.



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.

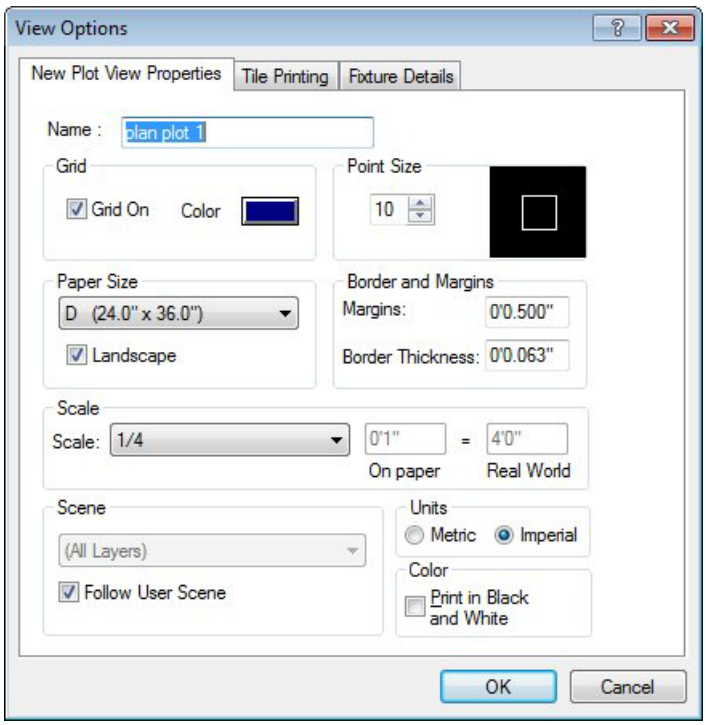
2 Right-click in the *New Plots* shortcut area, and then choose the type of plot that you want to create.

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*.

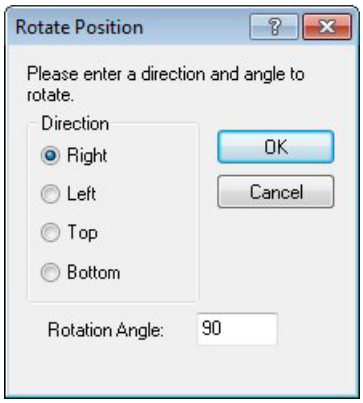
Step 3 - Plotting non-horizontal hang structures

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.

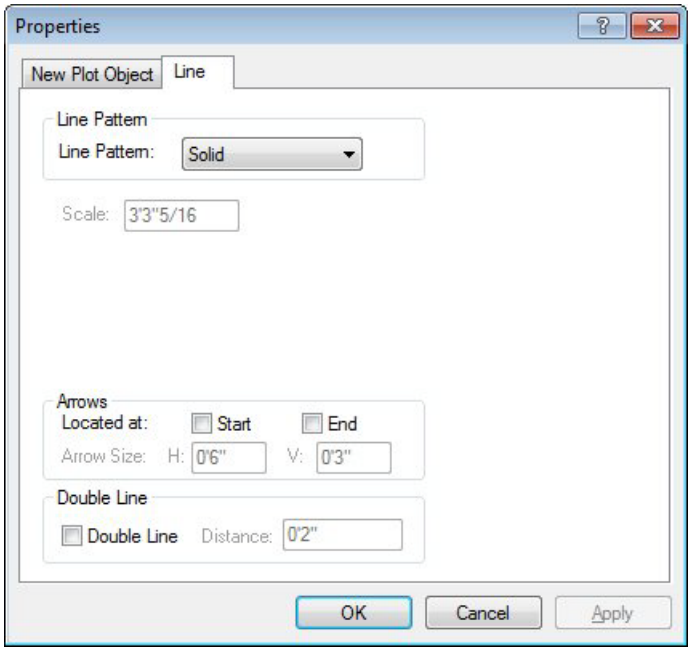
Tip: You can resize objects by clicking and dragging them.

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.

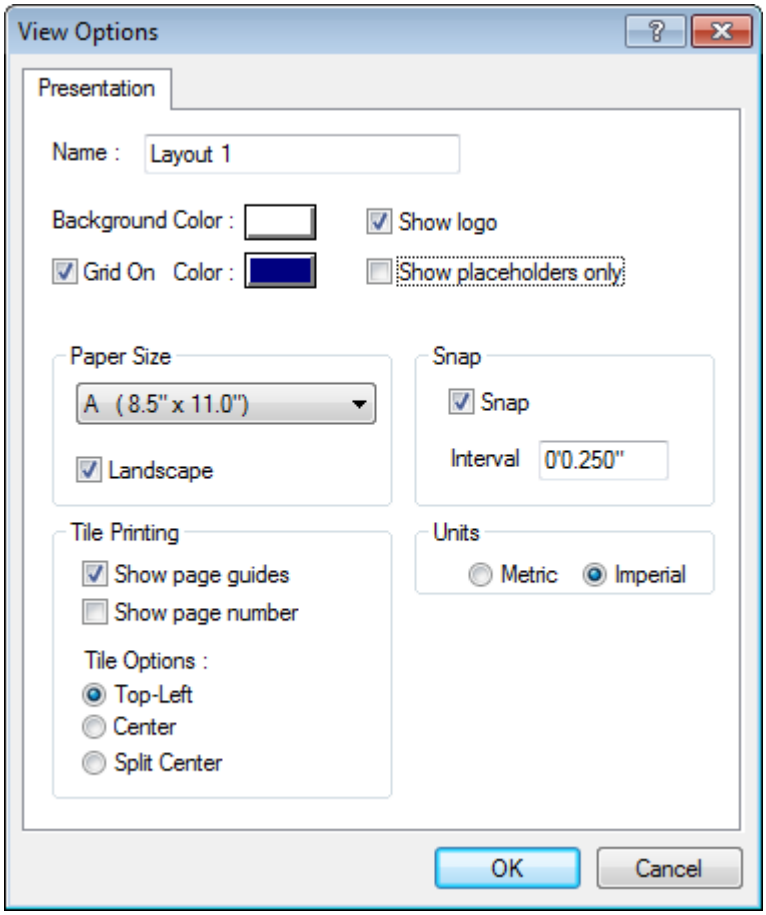
2 Right-click in the *Layouts* shortcut area, and then choose **New Layout**.

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.

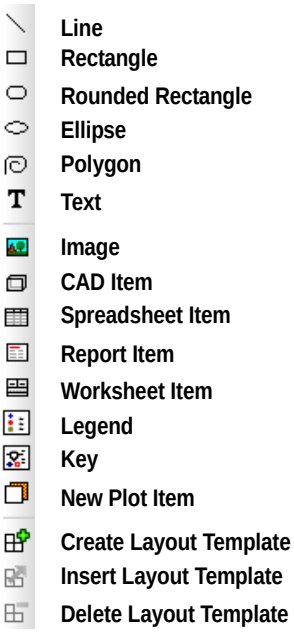
You may now begin placing objects on the sheet of paper. Use the *Pres Insert* toolbar to facilitate the process.

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 and 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.

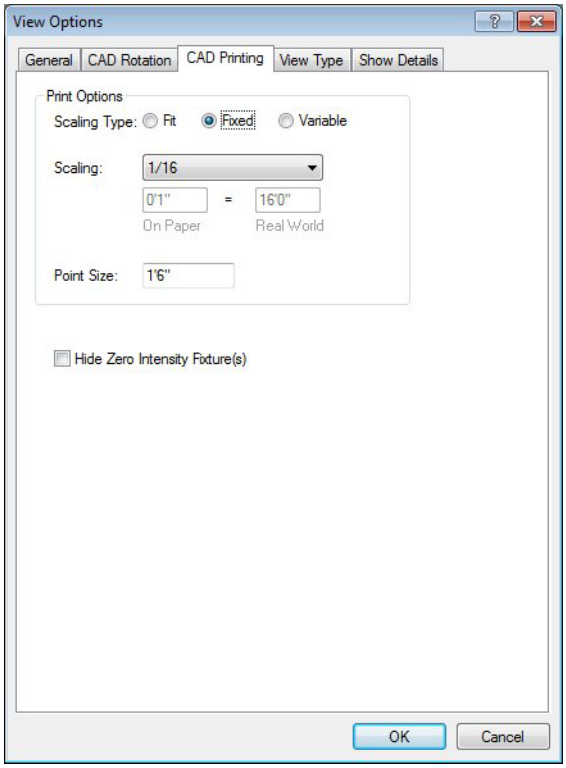
Step 3 - Modifying CAD items

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.

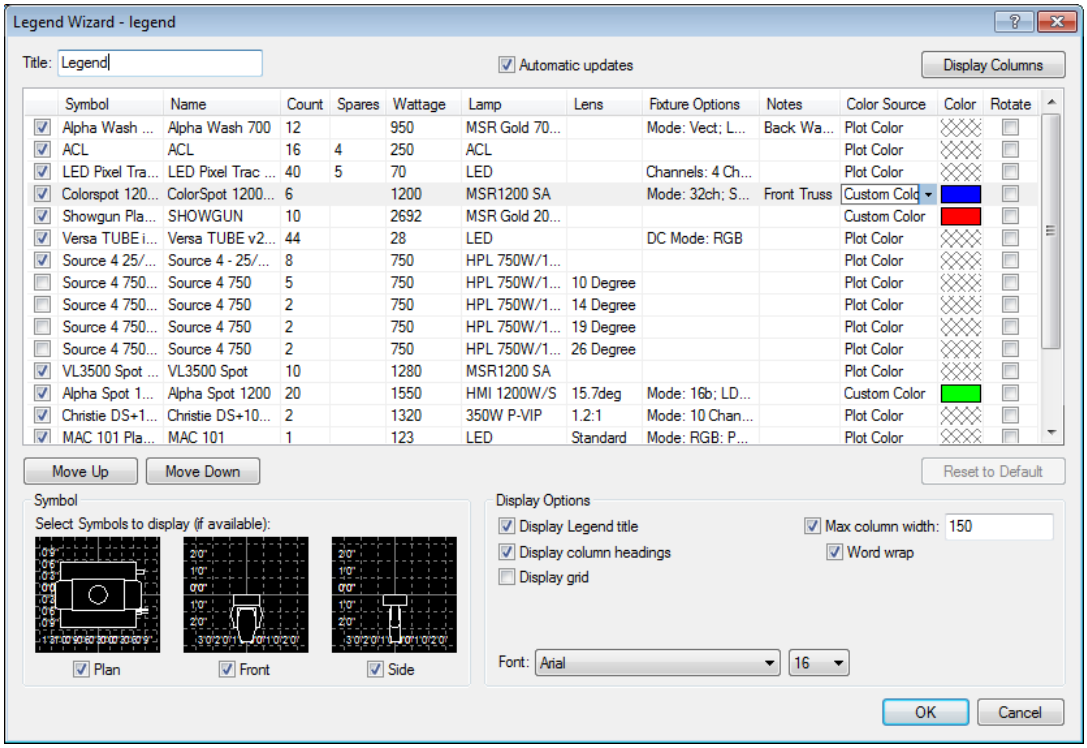
To create a legend

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*.

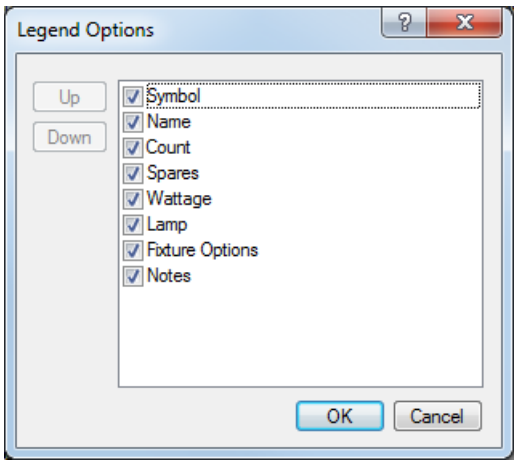
Result: The legend shortcut is stored in the shortcut bar and the *Legend Wizard* opens. The Legend Symbols Editor appears as displayed below. All fixtures in your CAD drawing are listed in the Legend Editor for possible inclusion in the Legend. All the options to configure your Legend are available from this dialog.



a. Legend layout

Options found here affect the layout of your legend.

- *Title*: The *Title* field will 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.
- *Automatic updates*: By default, this option is on and will automatically update the legend when there are updates in your CAD drawing
- *Display Columns*: The *Display Column* button is used to edit what columns are displayed in the legend. Clicking the button will bring up the *Legend Options* window. From here choose the columns that you want displayed by marking 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.



- Move Up: Select a row and use this button to move the fixture row up in the table.
- Move Down: Select a row and use this button to move the fixture row down in the table.
- Reset to Default: When a fixture row is selected that has cells that have been edited, this button becomes active. By clicking this button, the all the editable cells in the row will be reset to their default text.
- Symbol: The Fixture Symbol section displays up to three symbols that represent a fixture from different views, Plan, Front, Side, if available. Use the checkbox under each symbol if you wish it in the Legend.

Note: At least one symbol must be displayed for each symbol.

b. Legend information

Each symbol type that exists in your CAD drawing is displayed for possible inclusion in the legend. The counts of each symbol type is updated automatically as symbols with the same symbol type are added or deleted from the CAD drawing.

Options found here affect the symbol information that is displayed in the legend.

- *Display checkbox*: By default all fixtures are displayed in the legend, as identified by the check box. Click to deselect this check box if you do not want the highlighted fixture in the table to be displayed in the Legend.
- *Symbol*: This column displays the name of the Symbol from the Library which represents the fixture in your CAD drawing and will be displayed for this fixture on the Legend.
- *Name*: By default, the name of the fixture will be displayed, as listed in the Library Browser. If you wish to edit the displayed fixture name, click in the cell and type in your preferred Name for display.
- *Count*: By default, the total number of the fixture in the drawing appears in the Count column. This value includes only hung fixtures in your show file. If you wish to edit this number, click in the cell and enter a new count.
- *Spares*: By default, this column is blank. If you have spare fixtures that you would like to display on your Legend, click in the cell and enter a number of Spares.
- *Wattage*: By default, the table displays the wattage for the Lamp in volts. If you wish to edit this value, click on the cell and enter a new Wattage value.
- *Lamp*: displays the lamp of the fixture.
- *Fixture* Options: displays any options that are available on the fixture. To edit this cell, click on it and add or remove any of the text.
- *Notes*: Type in any additional notes that should be displayed for this fixture in the Legend.
- *Color Source*: By default, the Color Source will be set to Plot Color, which means the Legend will automatically display the symbol using the Layer Color or Object Color of the fixture, if the color is the same for all instances of the fixture in your drawing. If the color is not the same, the Plot Color will automatically be set to Black. Alternatively, you switch this column to Custom Color, and then use the column beside it using the colorbox to select a custom color for the symbol.
- *Color*: Click the Colorbox to open a color picker. Select the new color that you want to apply to the symbol, and then click OK
- *Rotate*: By default the checkbox is unchecked to display the fixture symbol in its default orientation. To rotate the symbol, click on the checkbox and select it.

c. Legend display options

How a legend appears in a report can be customized to meet your needs.

Options found here affect how a legend appears in the report.

- *Display Legend Title*: Enable this option to display a title for your legend.
- *Display column headings*: Enable this option to display the headings of columns in the legend.
- *Display grid*: Enable this option to show a grid separating symbols in the legend.
- *Max column width*: The number sets the width of the columns. The width must be set in order to set Word Wrap option. (Note: The minimum width for a column is 30).
- *Word wrap*: Enable this option to wrap text that exceeds the max column width.
- *Font*: Use the drop-down menu to select a support font style
- *Font Size*: Set the font size for all text in the legend layout.

To create key items

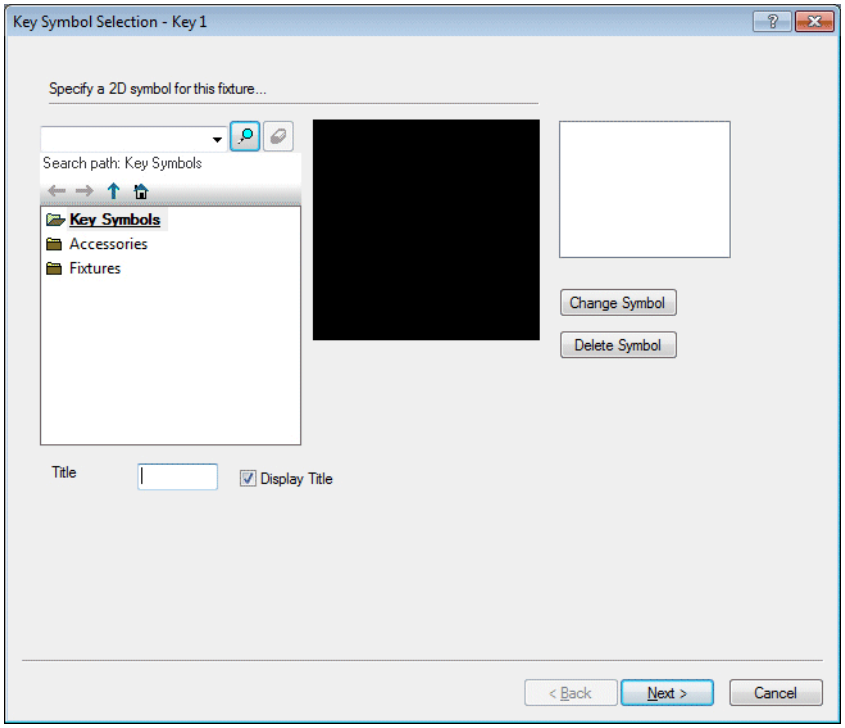
1 Right-click in the *Wizards* shortcut bar, and then select *New Key*.

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



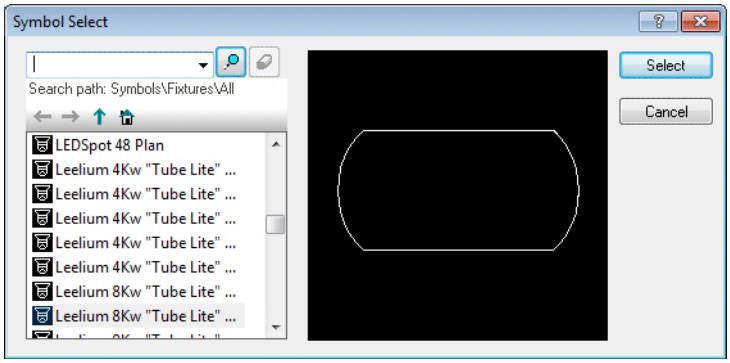
Options in Step 1 allow you to choose the symbol that will represent the fixture or accessory in the key.

- *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.

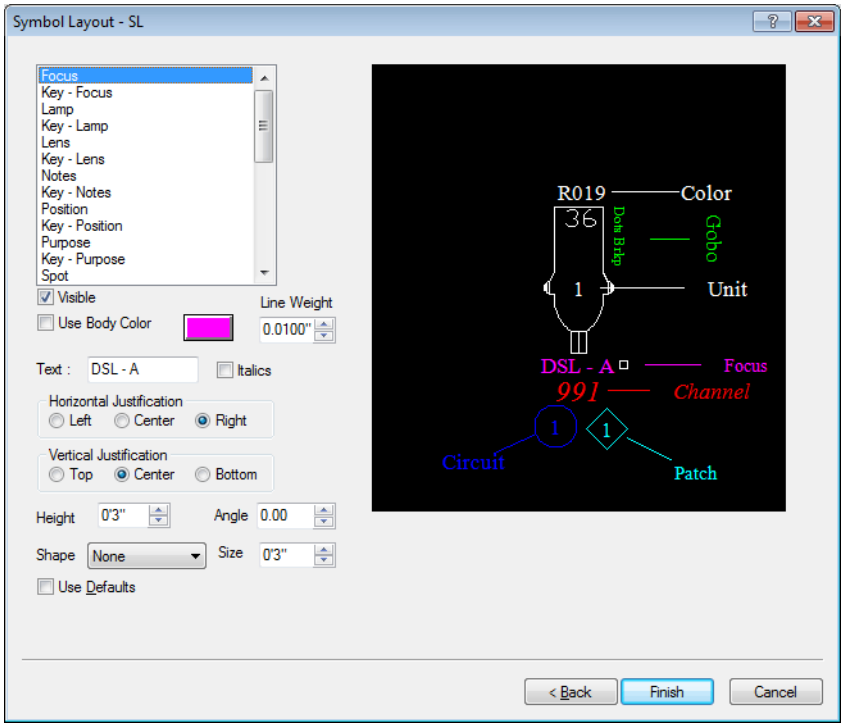
Changing symbols

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.

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.

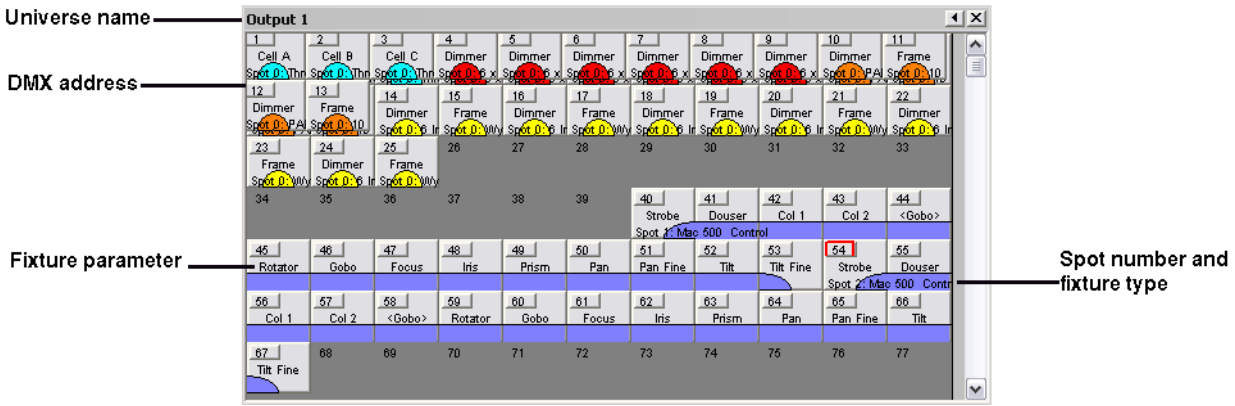
Each box in the patch universe represents one DMX channel. Fixtures are represented by colored bumps that span the number of required DMX channels.

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.

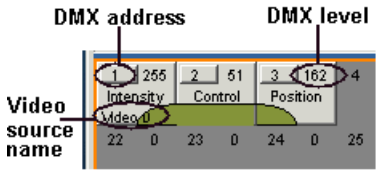
The fixture parameter controlled by each channel is identified in the center of the box.

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



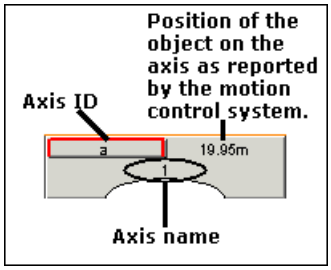
Reading the patch for streaming video sources

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

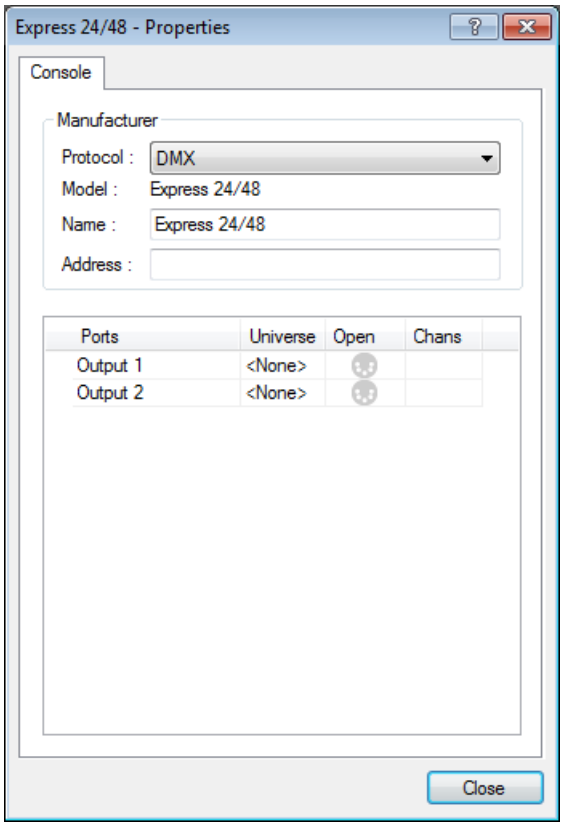
To insert a console

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 the *Universe* column to display a drop-down arrow, and then select the WYSIWYG universe to patch to the selected port.

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.

6 Repeat the above steps to bind all ports to their appropriate patch universes.

7 Click *OK* to close the *Properties* box.

8 Click *Connect*.

Result: The status of the console changes to “connected.”

9 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

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) 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.

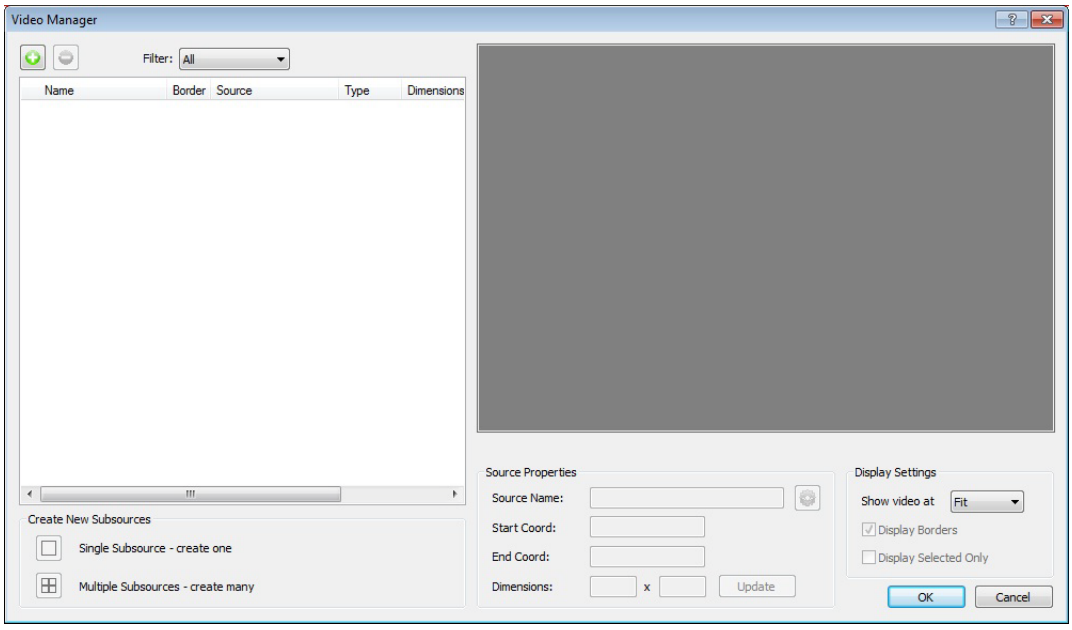
Notes: If you cannot view these file types, you may not have the proper CODEC installed on your PC. Install the appropriate CODEC, and then try viewing the file again.

Video sources can be split it up into subsources, which enables you to apply portions of the video to multiple screens, 3D primitives, LED Walls, or projectors. For details on splitting video, see the *WYSIWYG Reference Guide*.

Once you configure the video source and attach it to an object, you use the Video Designer Tool in Design mode to play the video.

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 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 in the *WYSIWYG Reference Guide*. (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.

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*.

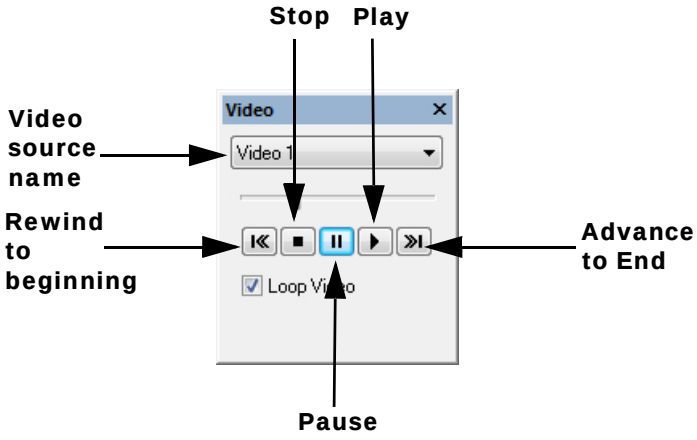
Step 3 - Using the Video design tool to play the video

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 51.

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 51.

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 68.

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 56.

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 56.
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 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*.

Lesson 14 - Using the 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 the *WYSIWYG Reference Guide*.

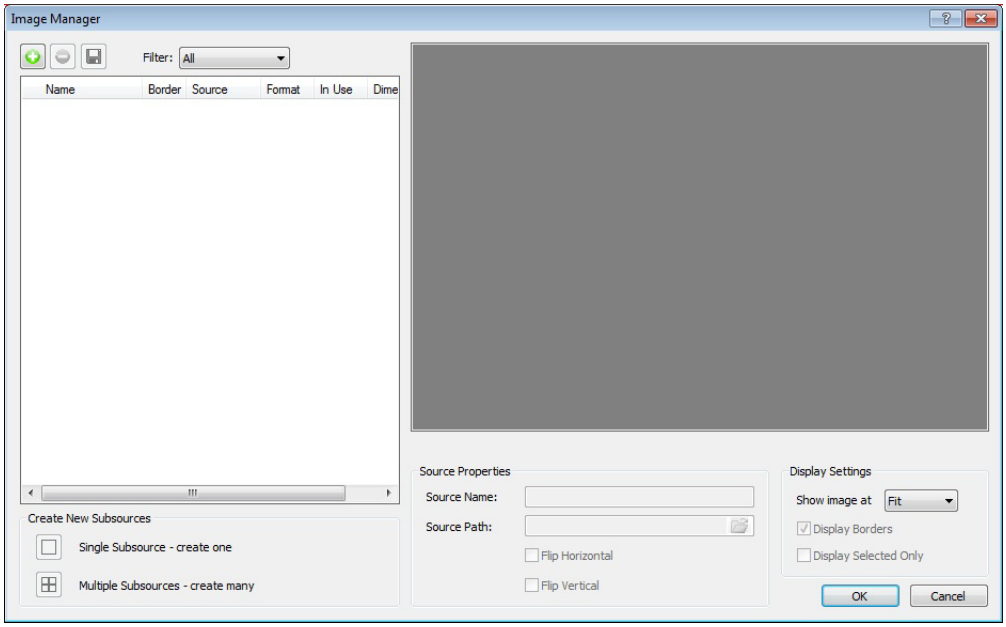
You can use the Image Manager to create subsources of an image (i.e., divide the image up), which can then be applied to objects such as surfaces, screens, LED walls.

Step 1 - Creating an image source in the Image Manager

To access the Image Manager

1 In CAD mode, select **Tools** > **Image Manager**.

Result: The Image Manager appears.

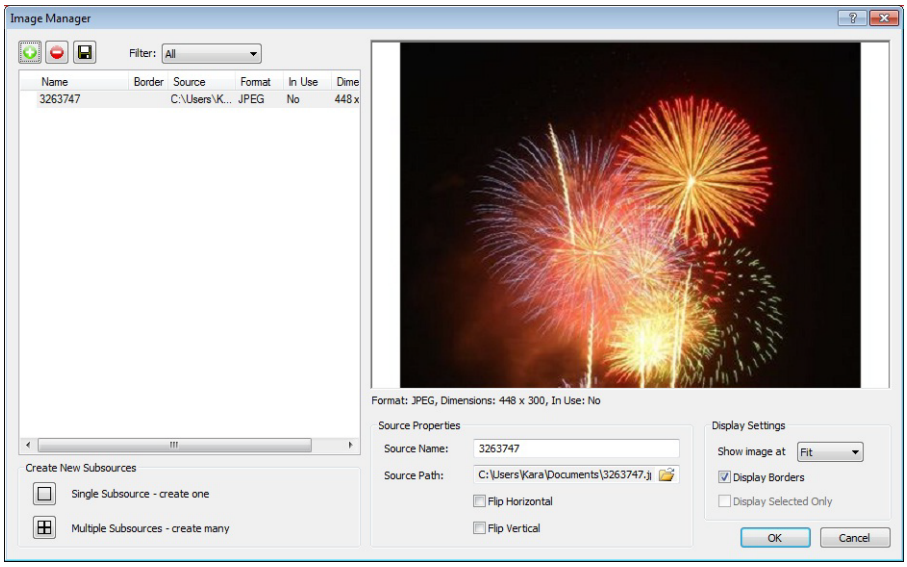


To create a new image source in the image manager

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* below. 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* below. 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% or 200%
  - *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*).

Step 2 - Flipping and exporting image sources

To flip an image source

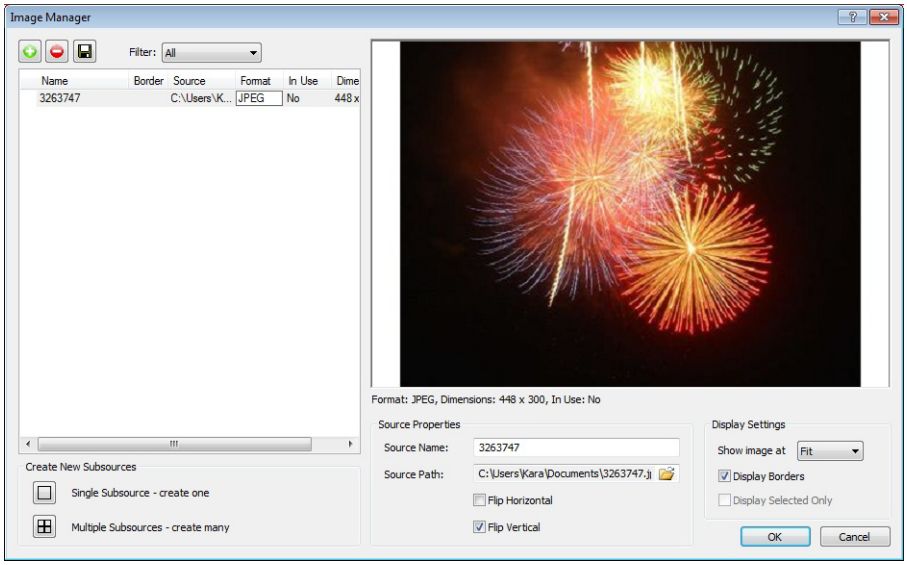
In situations where a surface was drawn in a clockwise direction, the texture may appear to be "flipped" either horizontally or vertically. To avoid this problem, flip the image source using the *Flip Horizontal* or *Flip Vertical* checkboxes, as shown below. Note that you cannot flip image subsources; 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.

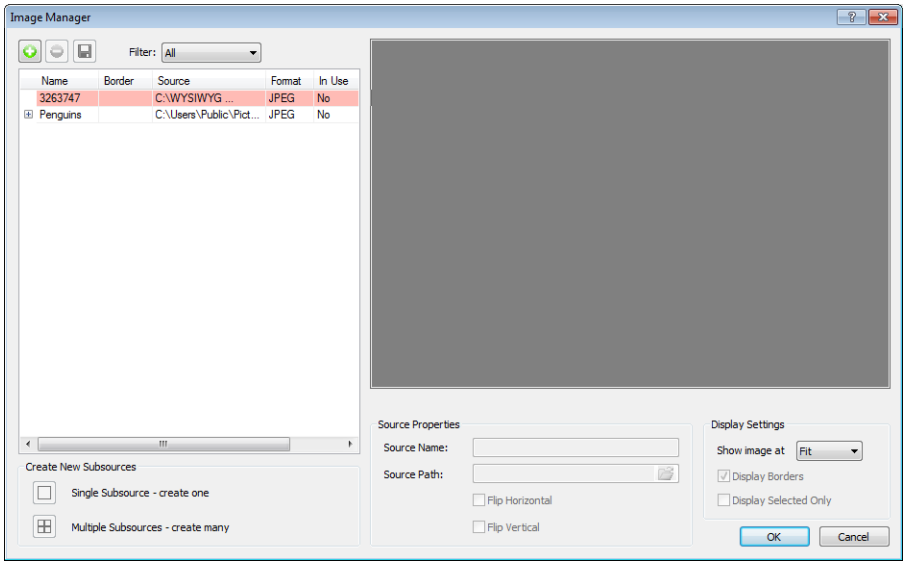
4 Type a name and select a file type for the new image, or accept the default file type of PNG.

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*.

Step 3 - Correcting 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*.

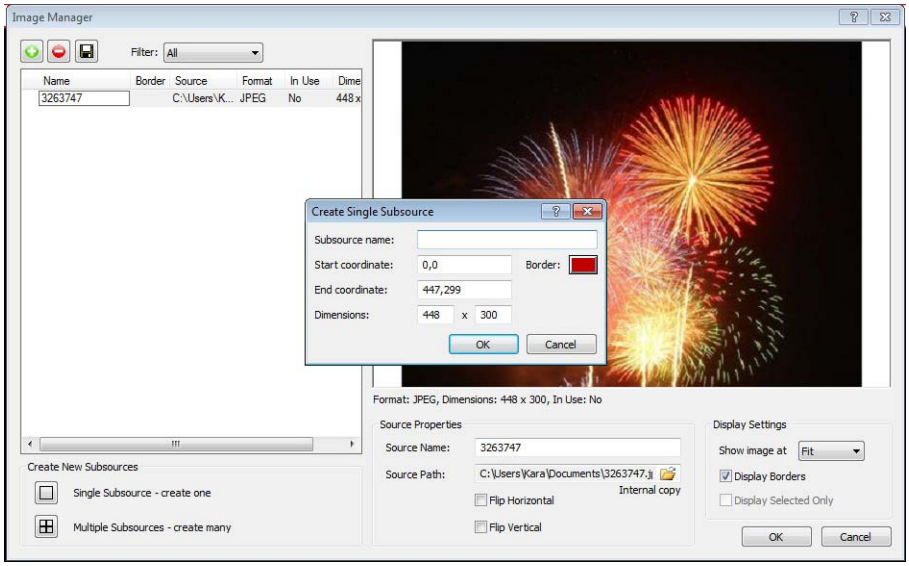
Step 4 - Creating image subsources

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.

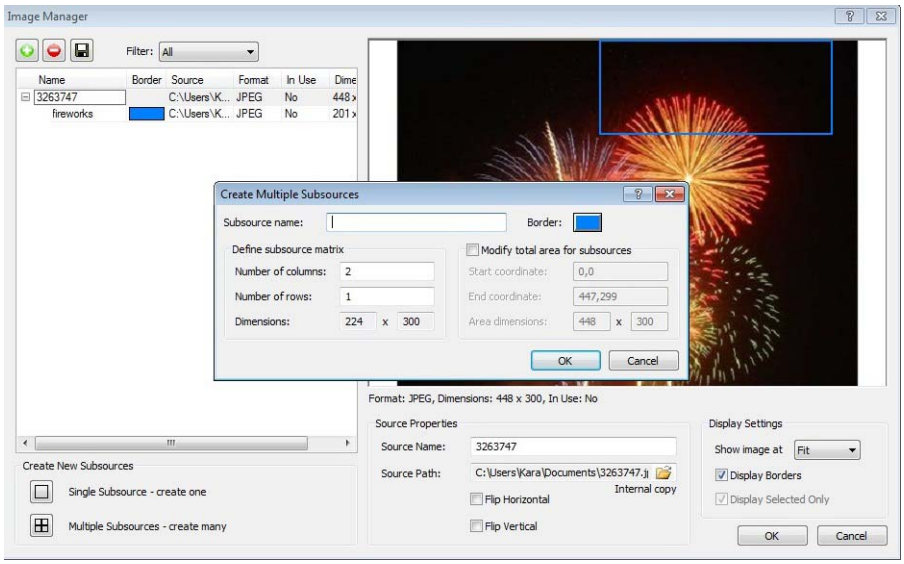
5 Click the Border box to specify the color of the border that will appear around the subsource when displayed.

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.

Step 5 - 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 the *WYSIWYG Reference Guide*.