Quick Start Guide

Product Release 2024 March 2024

Table of 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.

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

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

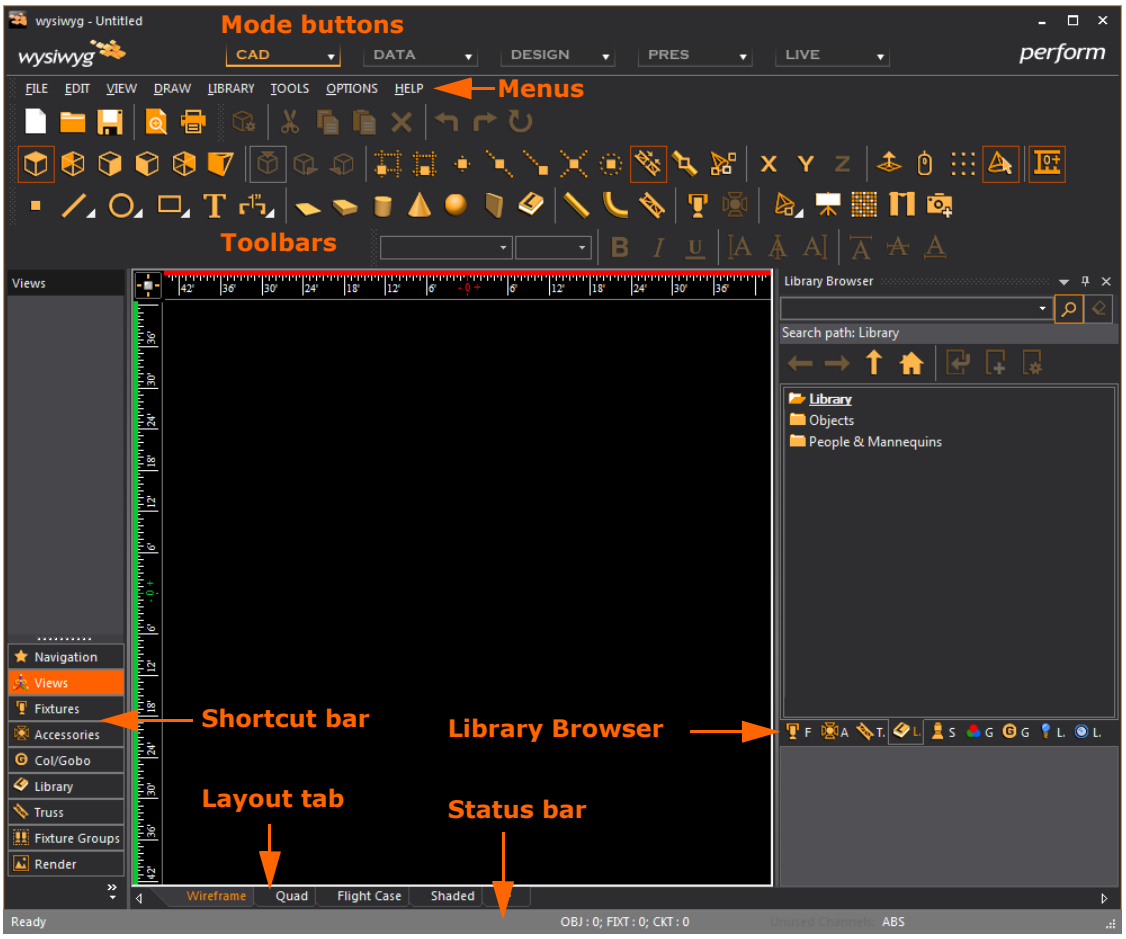
To start working in WYSIWYG

1 From the WYSIWYG Welcome window, choose one of the following 4 options:

- *New File* starts a blank new project file.
- *Open File* will display a browse window to select an existing .wyg file, or click on one of the recently used files shown under *Open File*.
- *Import File* allows you to import a file created from another application, which is saved in .dwg/.dfx or .skp file format.
- *Templates* lists all .wyt Template files which contain your file setting preferences. Select one of these files as your starting point for your project.

2 For the purposes of this tutorial, under *New* click *File*.

Result: The *WYSIWYG* main window is displayed.



Step 2 - Mode buttons

The Mode buttons located along the top of the user interface are used to access the different working modes available within the WYSIWYG levels. The currently selected mode is shown in orange, while modes not selected are shown as black. Clicking the appropriate mode button will change the view to that mode.

Note: Specific Layout tabs in any mode can be selected from the mode's drop-down menu, enabling rapid change of views easily.

Based on the WYSIWYG product that you have purchased, the modes available are some or all of the following:

- **CAD** - CAD mode is where you create your show drawings or "model." This includes drawing your venue, set pieces, lighting positions, focus positions, and lighting fixtures. This is also where you can import CAD files.
- **Data** - This is where all of your fixture information is compiled into spreadsheets. In this mode, you can view and edit the data for all your fixtures.
- **Design** - This mode provides lighting designers with the tools for experimentation. In Design mode, you can create and save static lighting Looks using the Designer tools, and then save and render those looks, all without having to patch fixtures or connect to a console. Additionally, you can integrate automation/motion control with your show.
  Note: (Design mode is only available in WYSIWYG Design and Perform).
- **Pres** - Presentation (Pres) mode contains all the tools necessary for creating professional printouts of your show documents/paperwork (i.e. reports and lighting plots).
- **Live** - Live mode is used for graphically simulating the output of a lighting control console or compatible offline editor. This is where you can pre-cue and pre-visualize your show. (Live mode is only available in WYSIWYG Perform).

Step 3 - Menu bar, toolbars

Below the Mode buttons is the Menu Bar. The available menus change as needed when moving from mode to mode.

Toolbars provide button access to most commands and can be used in lieu of the menu bar. Toolbars have a grab bar on the left that is used to move the toolbar around the screen. They can be placed on the edges of the work area (top, bottom, left or right), and can also be dragged off the edge of the work area and into their own window.

To open/close a toolbar, right-click in the toolbar area to display a menu of all the available toolbars. Enable the checkbox next to a particular toolbar to display it.

Step 4 - Layout tabs

Located at the bottom of the screen are Layout tabs. Each mode has designated layouts configured to the functionality and tools offered in each mode. Think of each layout as a workspace in your file for a different part of your project.

Step 5 - The Library Browser

The WYSIWYG Library is an essential component of WYSIWYG. The Library contains an extensive catalog of lighting fixtures and accessories, truss, gobos and gels, 3D objects, etc.

You can browse the contents of the library in the *Library Browser*. In the *Library Browser*. you can review each item's properties. When you find something you wish to use in your project, you can insert it from the library, or you can create a shortcut of the object, which will place it in the Shortcut bar for easier access

Step 6 - 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.

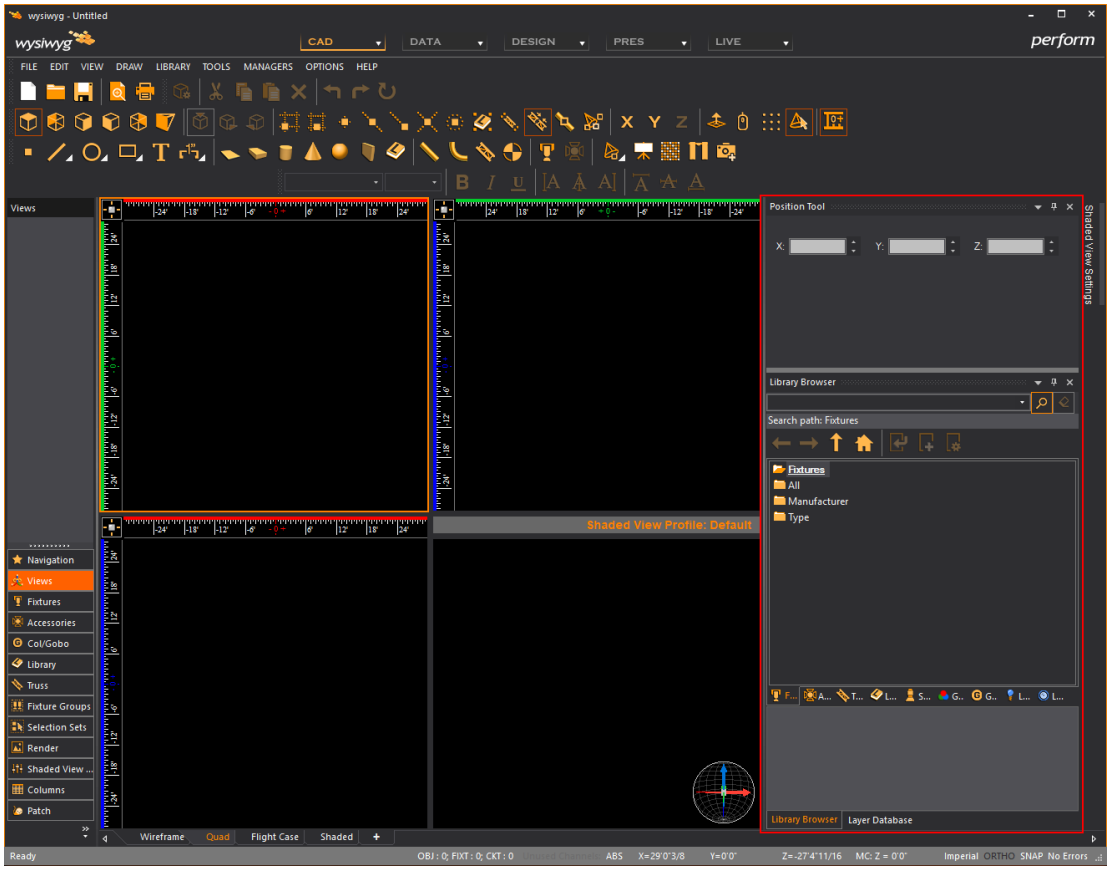
Example: You can save a navigation shortcut to CAD mode/Quad layout so that the next time you want to go back to that location, you can simply click the shortcut. The navigation shortcut will also remember which tools you have open.

To create a shortcut to CAD mode/Quad layout

1 In *CAD* mode, click the *Quad* tab to open the quad layout.

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

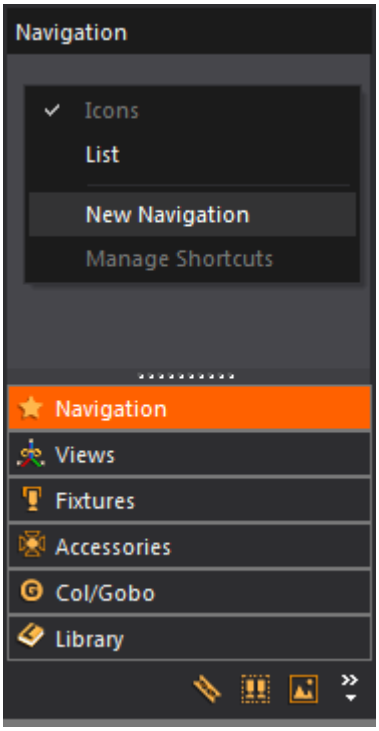
Result: The *Library Browser* now appears along the right side of your workspace.



3 Select the Navigation Shortcut bar.

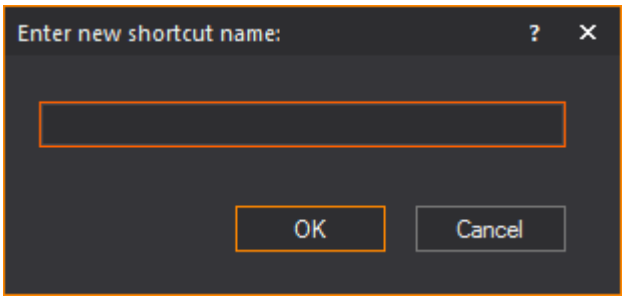
4 In the Navigation Shortcut bar, right-click.

5 Select **New Navigation** from the menu.

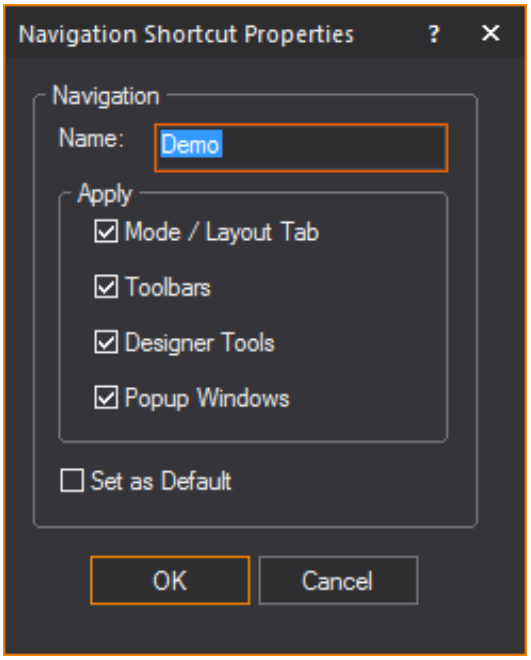


Result: The *Enter new shortcut name* window will appear.

6 In the *Enter new shortcut name* window, type a name for the shortcut and click *OK*.



7 The *Navigation Shortcut Properties* window will appear.



8 Click *OK*.

9 Close the *Library Browser* by clicking on the *X* button in the top right corner of the *Library Browser*.

10 Switch to a different mode / layout tab by clicking on any of the Mode buttons along the top of WYSIWYG.

11 Now, in the **shortcut bar,** under the *Navigation* section, click on *the shortcut you just created*.

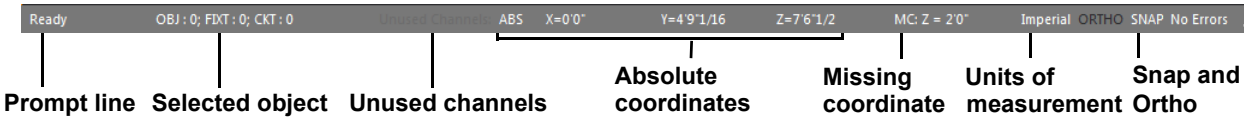
Result: This will bring you back to **CAD** mode / *Quad* layout. Since the shortcut was created with the *Library Browser* window open, it will be automatically displayed for you.

Step 7- Status Bar

The Status bar is displayed along the bottom of the WYSIWYG screen, below the Layout tabs. The Status bar displays useful information while working on your project, such as the prompt line, the selected object information, the missing coordinate, and the status of the snap, ortho and absolute coordinates commands. The Status Bar also tracks and displays the coordinates of the cursor.

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

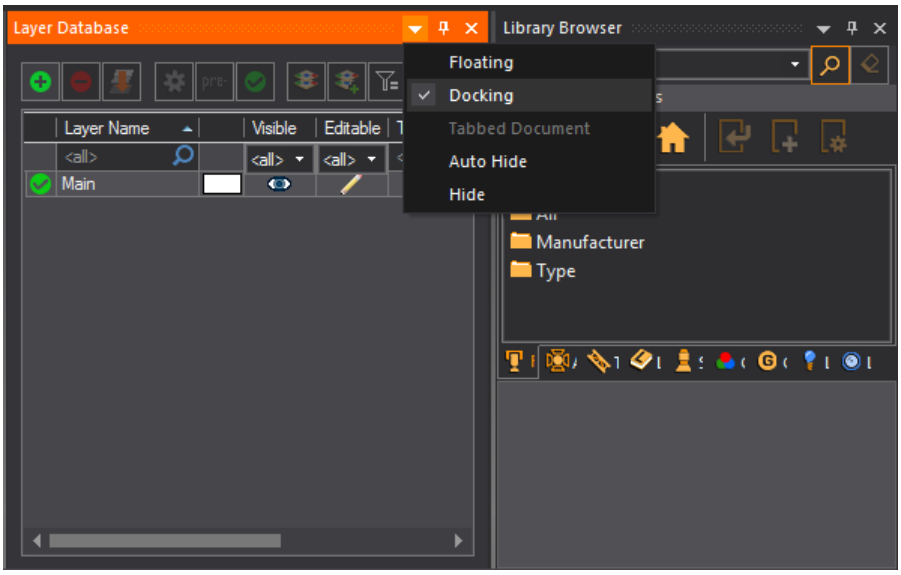
For more information on the Status bar, please refer to the Reference Guide.



Step 8 - Window Position Management

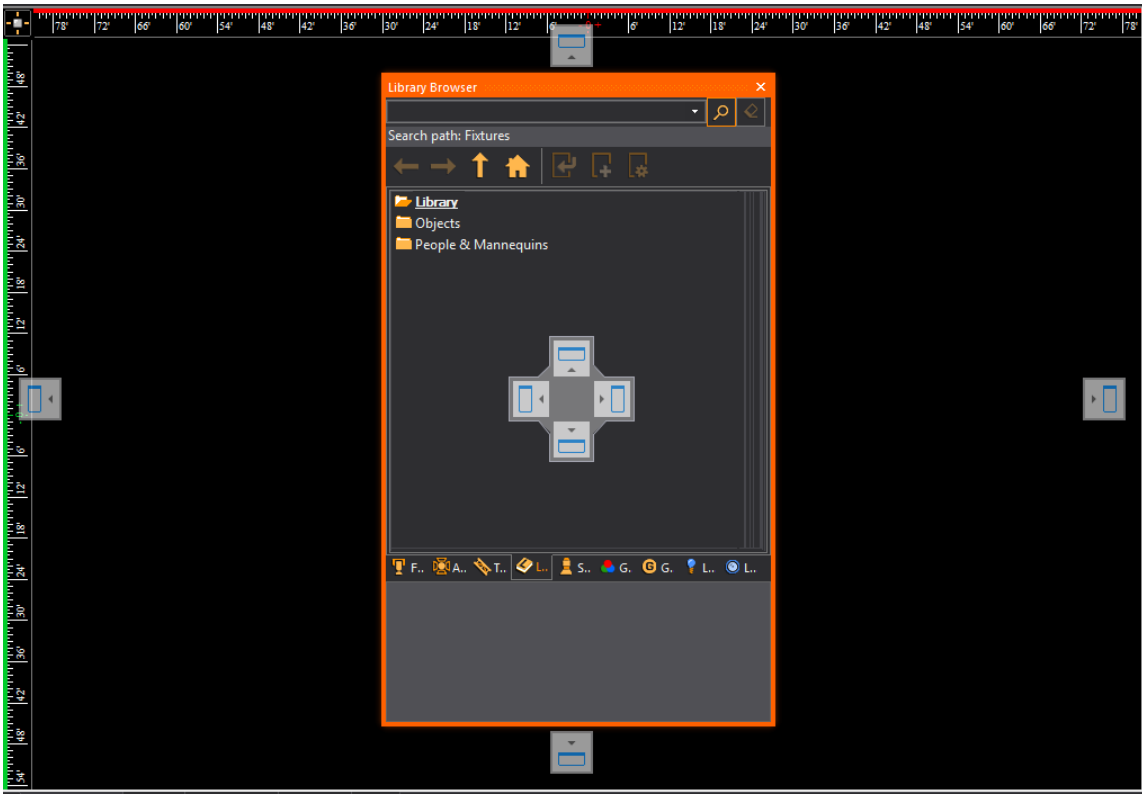
Most tools in WYSIWYG create a separate window where all the options of the tool are available. Window positions of these tools can be customized to suit most work styles. WYSIWYG will remember the window preference of the tool the next time.

Window position options can be accessed by clicking the down arrow at the top of the window, and opening the **Window Position** drop-down menu.

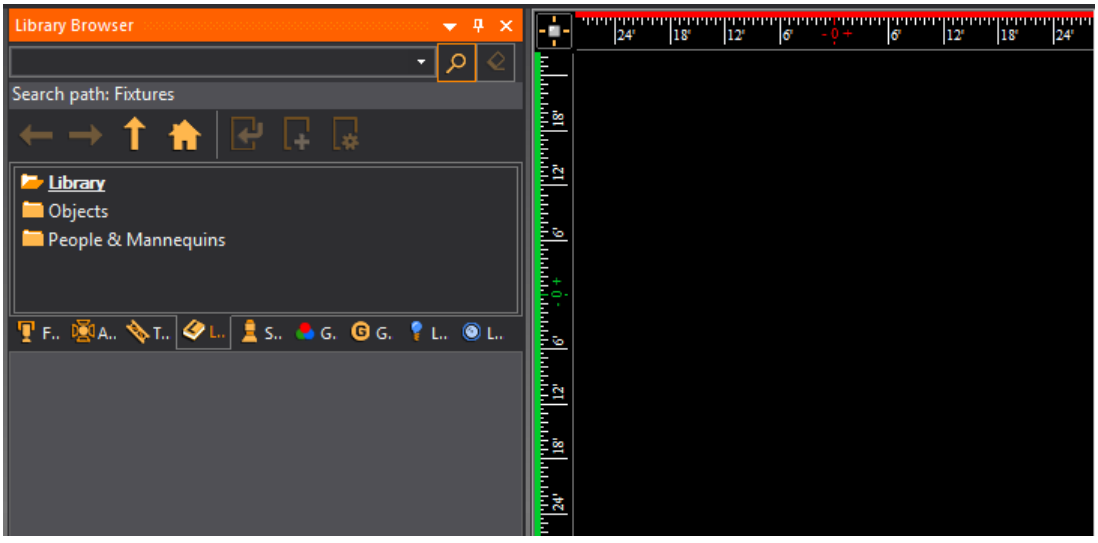


- **Floating** - The window will appear in the foreground of the WYSIWYG file. The window will not be attached to any border of the work space or any other window. If the resolution of WYSIWYG or the work space is

resized, the window will remain in its current position and in the foreground.

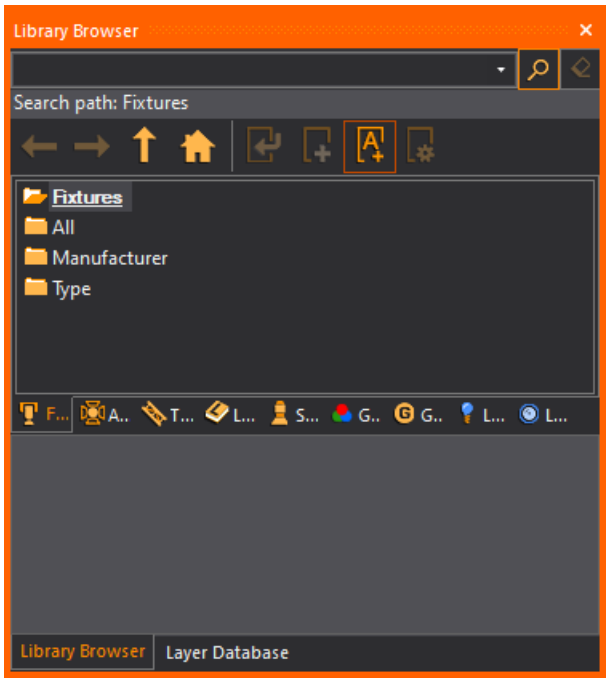


- **Docking** - The window will be attached to a border of the work space. If the resolution of WYSIWYG or the work space is resized, the window will change its resolution to match the new work space size.



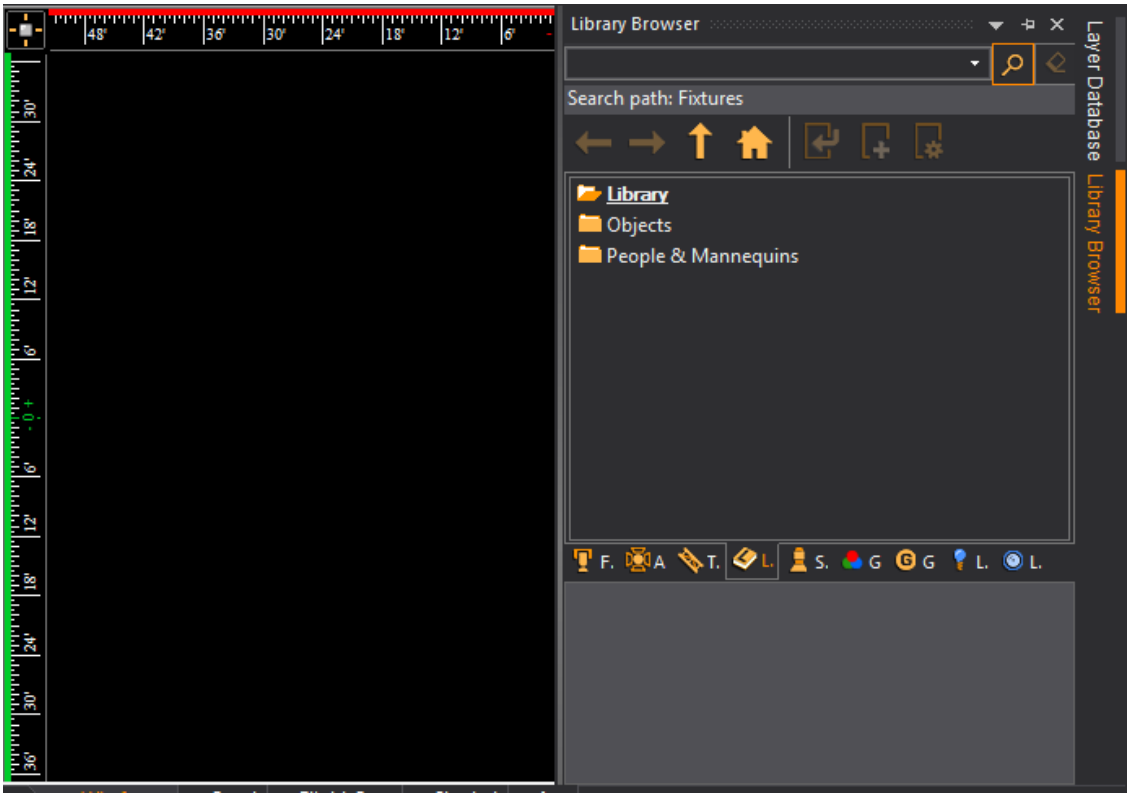
- **Tabbed Document** - Tools with similar display options can share the same window space. The tabbed document can be either floating or

docked. At the bottom of the window will be tabs showing the names of the different tools. Click on the tabs to switch between tools as needed.



- **Auto Hide** - This option is only available for docked windows. When enabled, the window will be hidden from view by default. A tab will appear on the side of the WYSIWYG application window with the name of the auto hidden window. When the tab is selected, the window will reappear in its previously docked location. The window will continue to

be visible until another window or feature is selected, or until you click off of it.



- *Hide* - The selected window will close. To view the window again, the feature must be accessed again.

To display a window as tabbed

1. Identify the tools/features that you wish to tab together.
2. Click and drag one tool window over to another tool window, until the arrows appear over the tool.
3. Without releasing the mouse, move your cursor to the square in the middle of the arrows that appear, and then release the mouse.

   Result: The tools will now be tabbed together.

To toggle the auto hide setting of a window

Click the *Pin* icon to toggle the auto hide setting of a window.

Step 9 - Layers

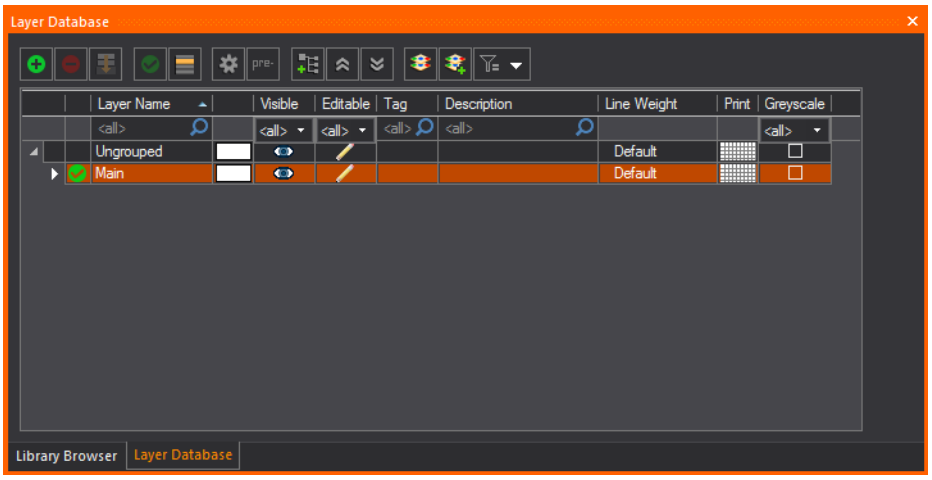
Layers are drawing aids intended to help you organize your plot. They are like transparent sheets upon which you can draw. In the same way that you can view several transparent sheets at once by placing them on top of each other, you can hide and unhide layers by choosing which sheets are in the stack.

A simple way to find out the layer to which any object in your plot is assigned, just hover over the object with your cursor in any of the wireframe views. A tooltip appears listing the object name and its layer. This is especially useful when you are working with multiple layers, some of which have the same color.

To create new layers

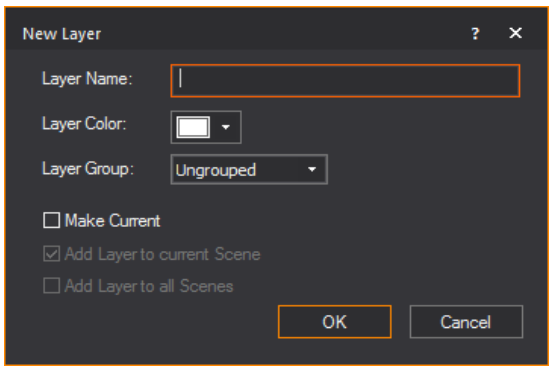
1 From the **Managers** menu, choose **Layers**.

Result: The *Layer Database* window is displayed.



2 To create a new layer, click the *New Layer* button.

The *New Layer* button.



3 Type a new name for your layer (for example, “Stage”).

4 Choose whether to add the layer to the current scene only or to all existing Scenes. (WYSIWYG maintains this setting for the next layer that you create.)

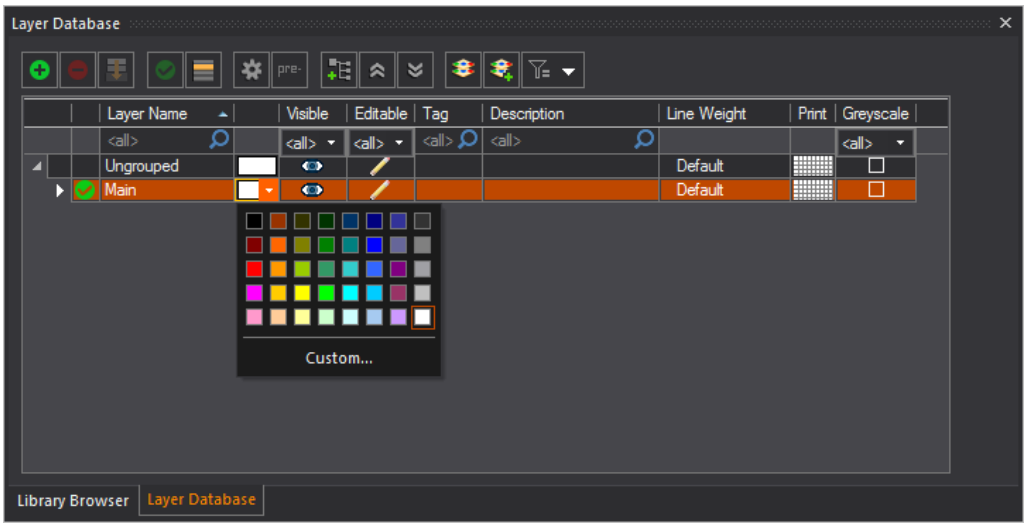
5 Click *OK*.

6 To make a layer and all objects on the layer visible on the plot and in shaded views, select the *Visible* icon. If not selected, the layer is not visible and is not, therefore, editable.

7 To make a layer and all objects on the layer editable on the plot and in shaded views, select the *Editable* icon. If not selected, the layer is not editable and is not, therefore, visible.

Tip: You might want to deselect a layer when you are finished working with it to avoid possible errors while working on other parts of the plot.

8 To change the color of the layer, click *Color Select*.



Note: It is recommended that you select different colors for each layer so as to easily identify the layers on the plot.

9 To change the line weight of the selected layer, choose the applicable thickness from the *Line Weight* drop-down box. If you leave *Default* selected, the line weight from the Show Options tab is applied to the selected layer. The line weight that you select applies to all objects on this layer (including library objects, pipes, and truss) and is visible in all wireframe views in all modes.

10 To view the properties of a layer (for example, name, color, and inventory of objects drawn on the layer), select the layer and click the *Layer Properties* button.

 The *Layer Properties* button.

11 Before closing the dialog, set your current layer by highlighting it, and then clicking *Set Current*. Alternately, you can set the current layer by double clicking in the first column on the left, beside a layer name.

 The *Set Current* button.

Note: The current layer is the layer that you are working on at the moment – any object that you draw is placed on this layer and assumes the layer's default properties when it is drawn. A check mark appears beside the name of the current layer.

12 To view and modify the scenes in which a layer is included, select a layer, and then click the *Scenes* button at the top of the Layer Database.

The *Scenes* button.

13 Click *OK* to close Scenes for Layers dialog.

Note: To quickly sort the layers within the layer database, click the appropriate column heading in the *Layer Database* dialog box. For example, click the *Editable* column heading to sort the layers by edit setting; that is, those layers marked as editable appear at the top of the list in alphabetical order.

Note: How columns appear in the *Layer Database* can be edited. The order of columns can be changed by clicking on a column header and dragging it left or right to the desired position. By right-clicking anywhere in the table area, a list is displayed which allows you to show/hide columns as you wish.

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

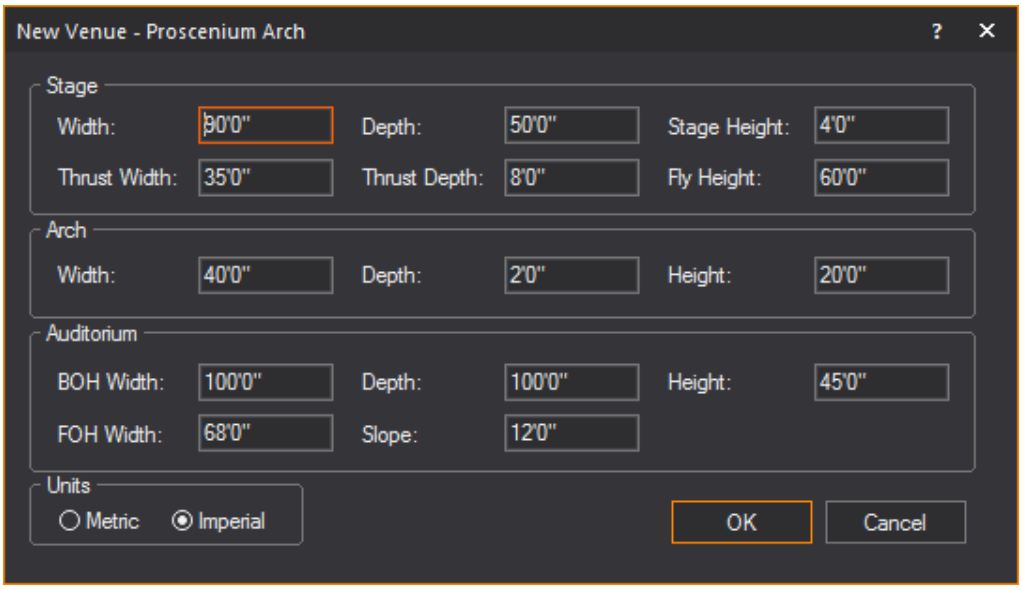
Step 1 - Inserting a venue

Begin in **CAD** mode *Wireframe* tab. This is the mode and layout tab in which you create your show drawings. In this example, you will add a Proscenium Arch Theatre.

To insert a venue

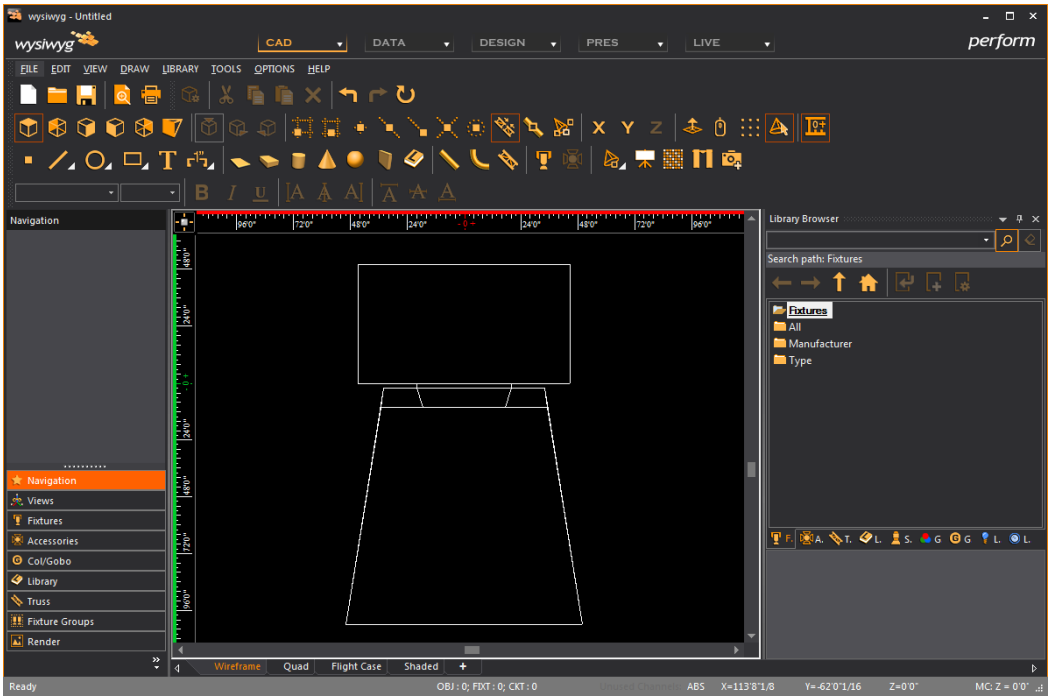
1 From the **Draw** menu, choose **Venue** > **Proscenium Arch**.

Result: The *New Venue - Proscenium Arch* window will open.

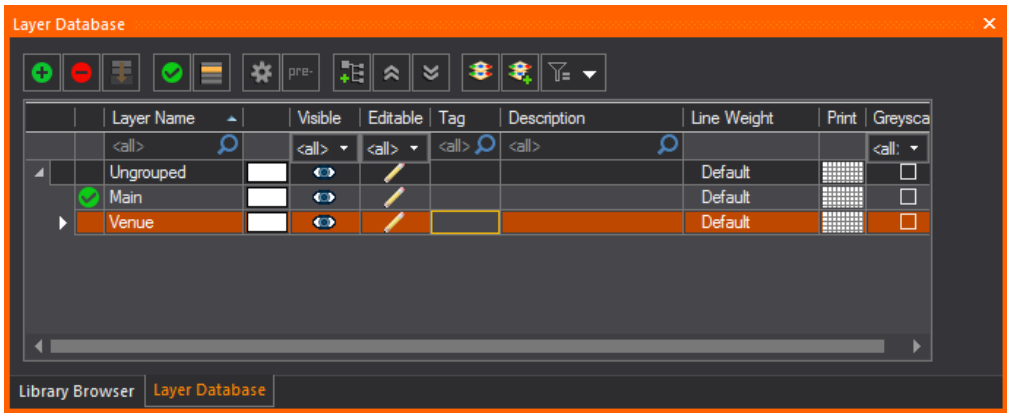


2 Accept the default values, and then click *OK*.

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



Note: Venues are automatically placed on their own "Venue" layer. The "Venue" layer is now listed in the *Layer Database*.



Step 2 - Coordinate System

Like any CAD software, WYSIWYG builds your model in a three-dimensional space. All objects in your drawing will be positioned at an X, Y, Z coordinate, which is the distance of the object from the Origin.

Using the Ruler which is displayed along the top and left side of the Wireframe view, you can easily find the location of the Origin.

In this file, the center of the Proscenium Arch at the Stage floor level is the Origin position.

Note: As you move the cursor around the Wireframe view, you will notice in the Status bar that the XYZ coordinate of your cursor's location is displayed. Notice that the Z coordinate is always 0 as you move your cursor. Since you are looking at the model from a top down view (called Plan View), moving your cursor left/right changes the X coordinate, and moving your cursor up/down on the screen changes the Y coordinate. Since you cannot change the Z coordinate by moving your cursor, you must set the Z Coordinate by setting the "Missing Coordinate".

Step 3 - Setting the Missing Coordinate

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

In the Status bar, notice that the Missing Coordinate (MC) is set to Z = 0. This means that any objects that you insert into your file (while working in Plan View) are placed at the X,Y coordinate you define, while the Z coordinate must be defined by setting the Missing Coordinate. The X,Y coordinate can be defined by clicking or typing in the coordinate values.

To change the Missing Coordinate, while the Wireframe view is selected (i.e. click in the view), hit the TAB button and a dialog appears to set your CAD coordinate for the Missing Coordinate. Let’s leave it at Z=0 for now.

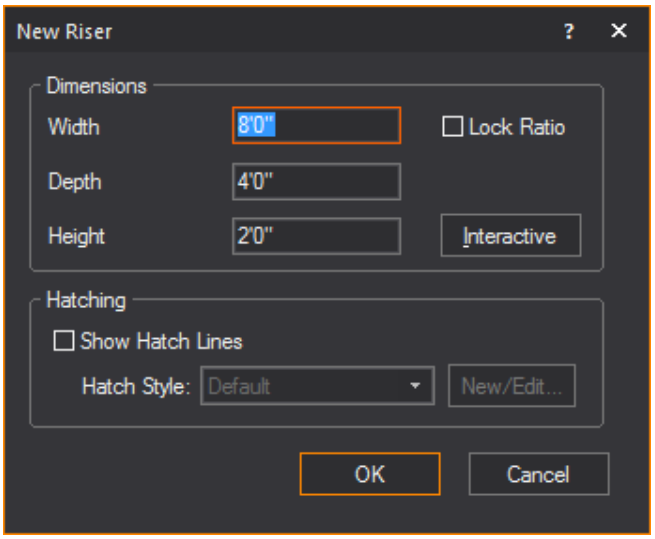
Step 4 - Building your set from CAD objects

A number of 3D primitives are available from the **Draw** menu such as circles, spheres, cylinders, and risers. In this example, you will insert a riser on the stage to have an elevated rectangular area for a drumkit.

To build your set from CAD objects

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

Result: The *New Riser* window will open.



2 The *New Riser* window, specify the dimensions of the riser.

Example: Width 10'0”, depth 10'0”, height 2'0”.

3 Click *OK.*

4 The riser needs to be placed in the center along the Y axis, leaving space in front of the riser for the other members of the band. Move your cursor in the Wireframe view above your stage area, right-click and select **Bottom Center** to insert the Riser from the center of its bottom surface, and click to drop the riser into your drawing.

Step 5 - Using the Command Line

The Command Line is an area in WYSIWYG where you can enter coordinates for the purpose of placing or editing objects in a document. Objects may be placed quickly and with precision by using the Command Line.

Coordinates are specified as X, Y, Z.

In our previous step, we want the riser’s center to be aligned with the Proscenium Arch's centreline (and maybe its bottom edge to be 4' upstage of the Proscenium Arch’s plaster line). We should remove the previous riser and make a new riser at coordinates “**0,9,0**”.

Using the Command Line to position an object

1. Select the Riser you inserted previously by clicking on one of its lines in the Wireframe view.
2. Click the DELETE button on your keyboard, or right-click on it and select **Delete**.

   Result: The riser will be deleted.
3. We will now redraw the riser. From **Draw** menu, select **Riser** and enter the dimensions like you did before, then click OK.
4. This time, rather than clicking with your mouse to define an insertion position, start typing "-5, 4, 0" and hit ENTER.

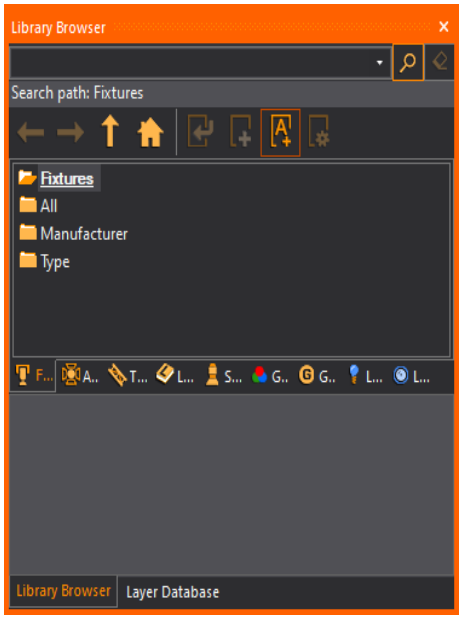
   Result: The riser will be positioned at exact coordinates of "**-5, 4, 0**"

Step 6 - Inserting library items

Use the items in the WYSIWYG library to build your project. In this example, you will add Mannequins with Library Objects to place a band on the stage.

To add Band members from library items

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



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

 The *Library* tool button.

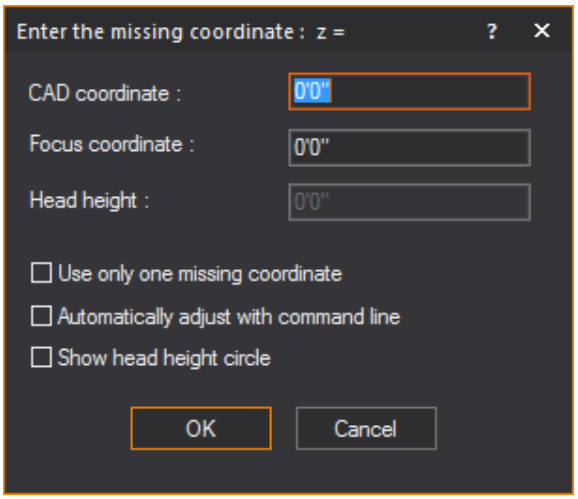
3 Navigate to the desired item, in this case, **People & Mannequins > Mannequins > With Objects**, select a mannequin with an instrument you wish to add (e.g. Guitar or Mic Stand), and then double-click to insert.

Note: To end the insert, either right click and select **Finish Insert**, or press the ESC key on your keyboard.

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 Insert a **Person with Microphone**, **Person with Electric Guitar**, and a **Person at Keyboard** on the Stage.

5 Now let's add a Drummer with Drumkit on top of the Riser which was drawn previously. Press TAB to display the *Enter the missing coordinate* dialog box.



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

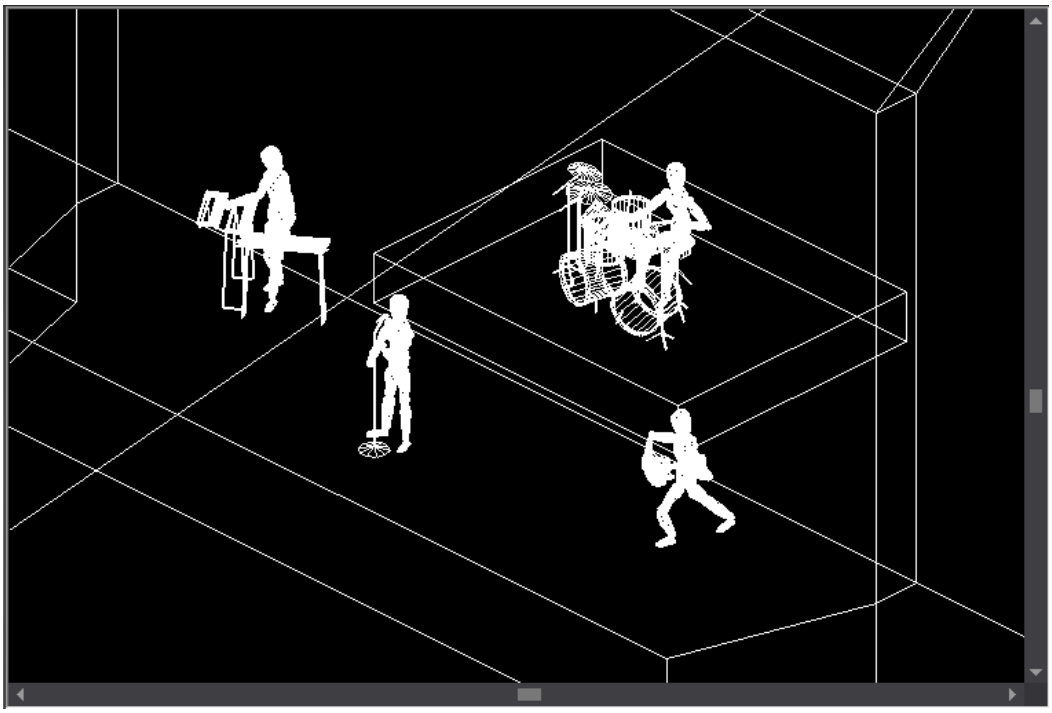
Tip: As you move your cursor the coordinates are displayed in the bottom right corner of the screen.

7 Insert the **Person at Drum Kit** on top of the riser.

Result: The **Person at Drum Kit** will be on top of the riser.

8 To get a better view of the stage, and make sure the **Person at Drum Kit** is on the riser, we will change our view of the stage. In the CAD Options toolbar click the *Isometric View* button.

The *Isometric View* button.



Result: We can clearly see that the **Person at Drum Kit** is on the riser from this view.

Step 7 - Building your set using CAD tools

WYSIWYG is equipped with CAD tools which allow you to draw three-dimensional models from scratch. In this example, you will draw a a simple backdrop that will be lit with a cyc to illustrate some of these tools.

To draw objects using the CAD tools

We will first draw a line. We will later turn the line into a surface to act as a backdrop.

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 this example use, **-20,20,0**), and then press ENTER.

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

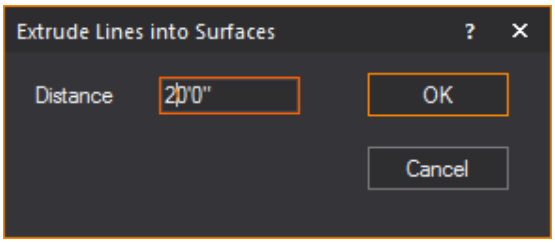
3 Specify the coordinates for the end point of the line (for example, **20,20,0**), and then press ENTER.

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**.

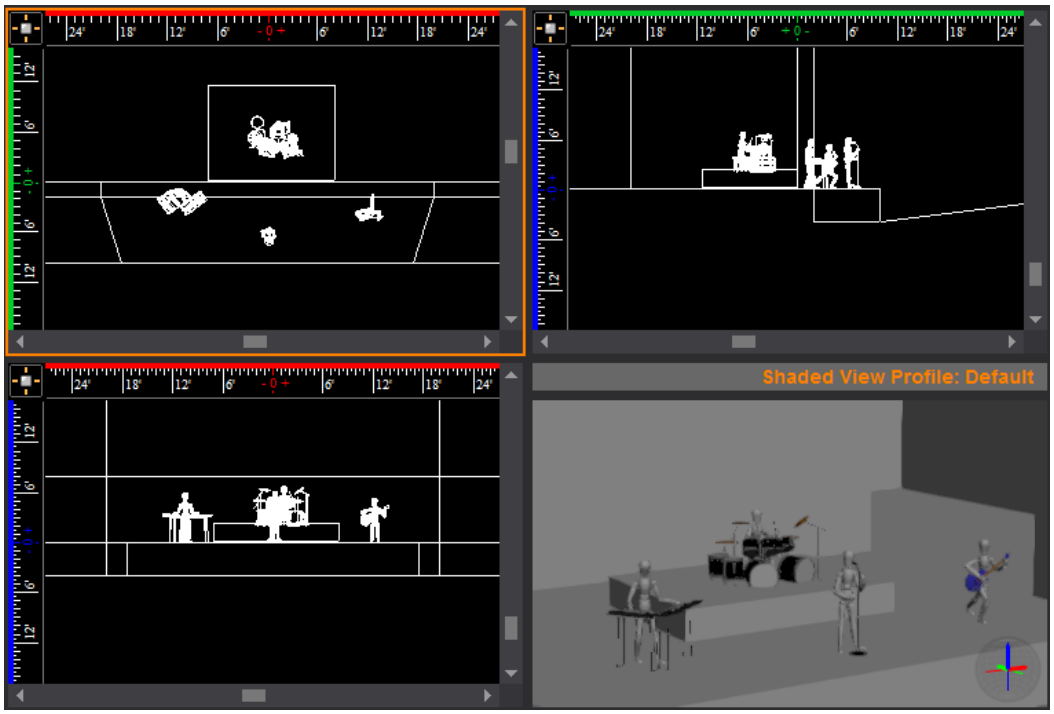
2 The *Extrude Lines into Surfaces* window will appear.



3 Enter the value for the extrusion distance, **20'**, and then click *OK*.

Result: The line is stretched into a surface.

Switch to the Quad layout tab, by clicking on the *Quad* tab along the bottom, or click on the down arrow at the right side of the *CAD* mode button and select *Quad*.



The *Quad* tab divides the workspace into 4 quadrants which display your stage from different viewpoints. The top left quadrant displays in a Plan View (top) by default; the top right in a Left View; the bottom left in a Front View; and the bottom right display is a Shaded View, which is an Open GL representation of the model.

Using the Quad tab, you can view your file from numerous perspectives all at the same time.

Step 8 - Customizing colors and textures

You have designed a simple stage with some band members, but everything is very boring and grey. Let's add some color and textures in the next steps.

Switch to Quad tab, which offers Wireframe views where you can select the object you wish to edit, and the Shaded view in the bottom right quadrant is available to visualize the color and texture properties.

To add color to the riser

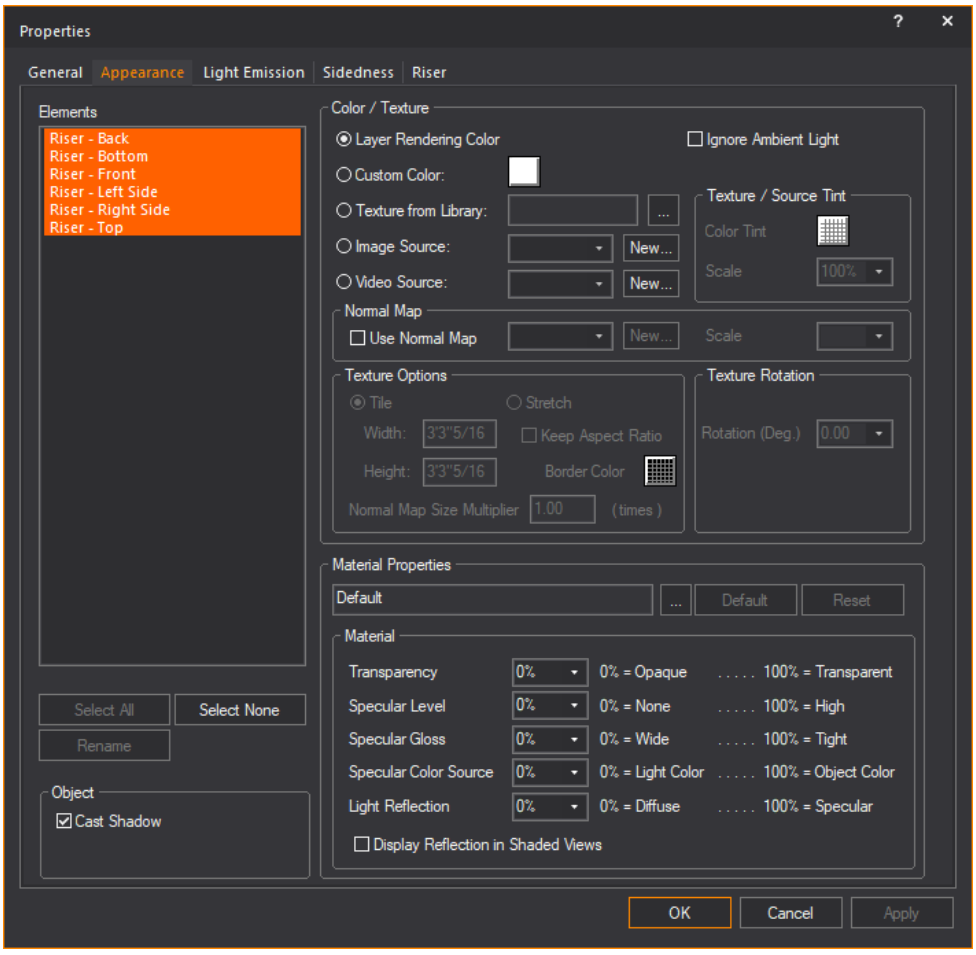
1 Select the riser by double clicking on it.

Result: The *Properties* window will open.

2 In the *Properties* window of the riser, switch to the *Appearance* tab.

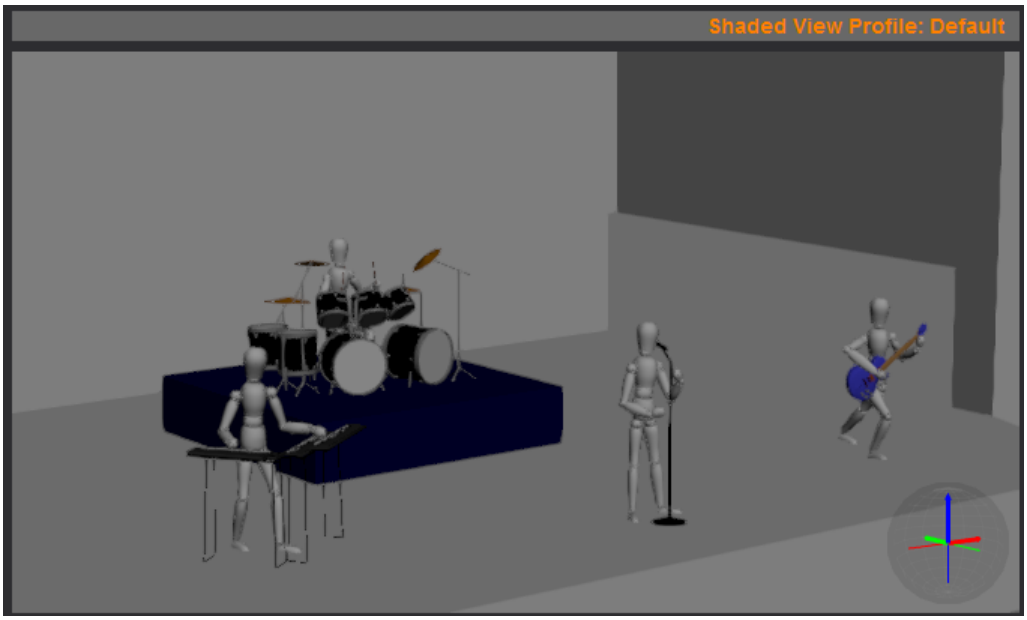
3 In the Elements list along the left side, you will find listed the 6 sides of the riser.

4 Highlight all 6 of the elements, and click on the color picker beside "Custom Color". Pick a color for the riser, e.g. blue.



5 Click *OK*.

Result: The riser now appears blue in the Shaded view.

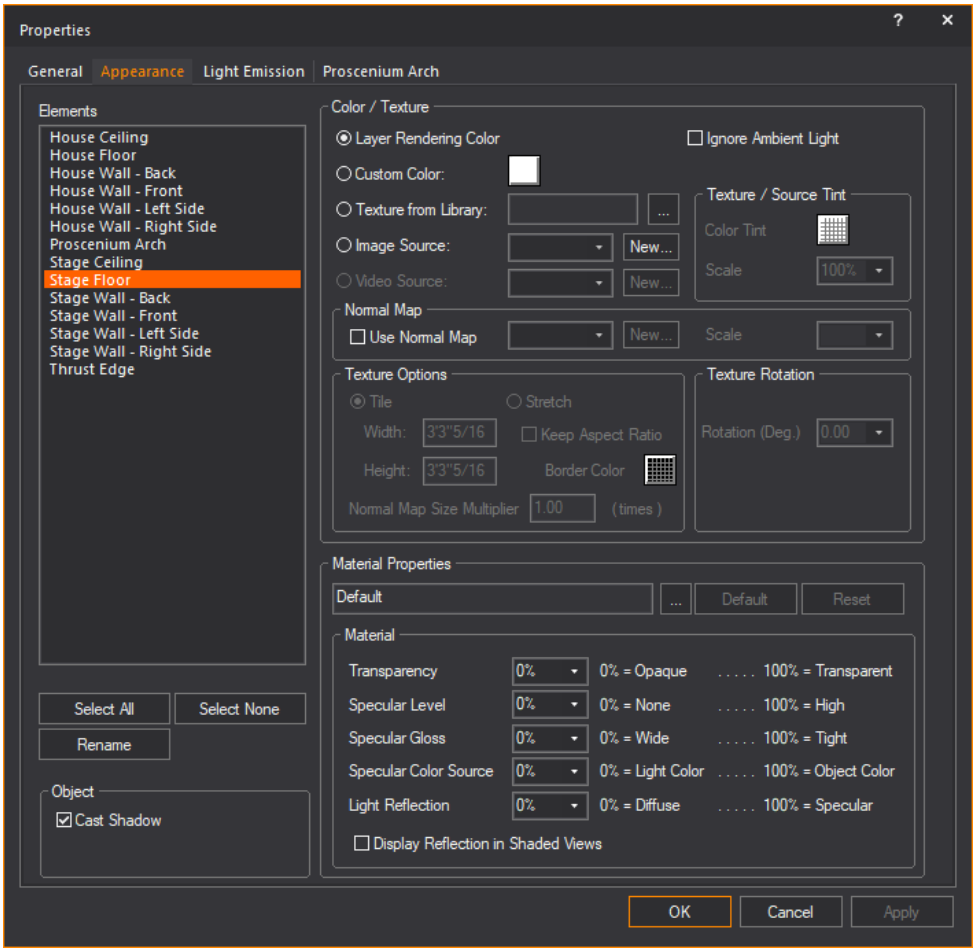


Now, let's add a texture to the Stage Floor.

To add color and textures to the venue

1 Select the Proscenium Arch venue, by double-clicking on any of the lines of the venue object.

Result: The *Properties* window of the Proscenium Arch will be opened.



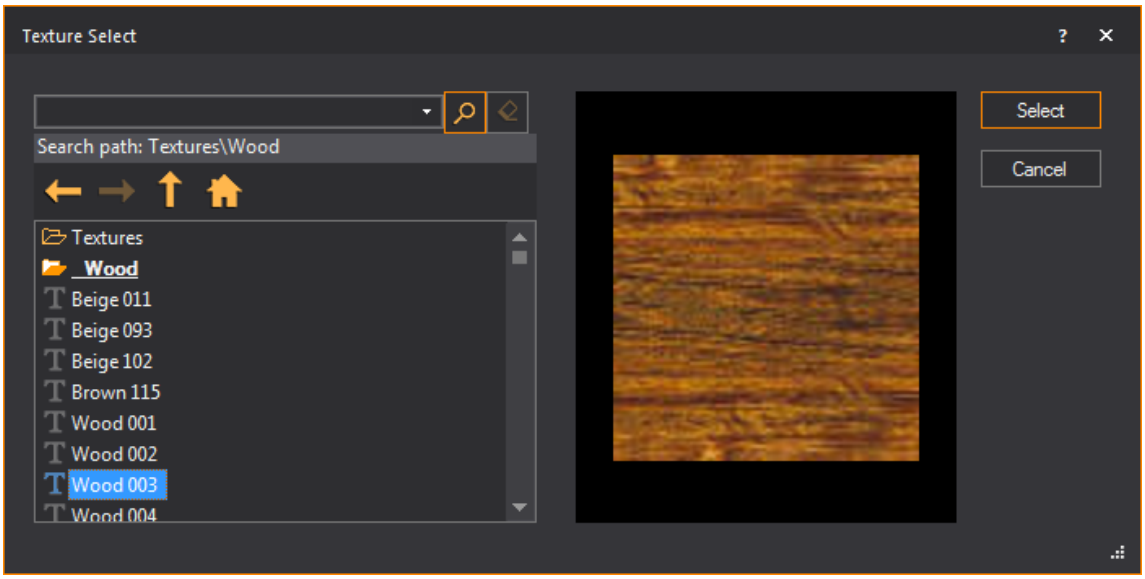
2 In the *Properties* window of the Proscenium Arch, switch to the second tab, named *Appearance* tab.

In the Elements list, all the different elements of the venue object are listed (which means you can individually set different properties for each element).

3 For our example, we will only select the **Stage Floor** element.

4 This time, select the radio button beside *Textures from Library*.

Result: The texture select window appears.

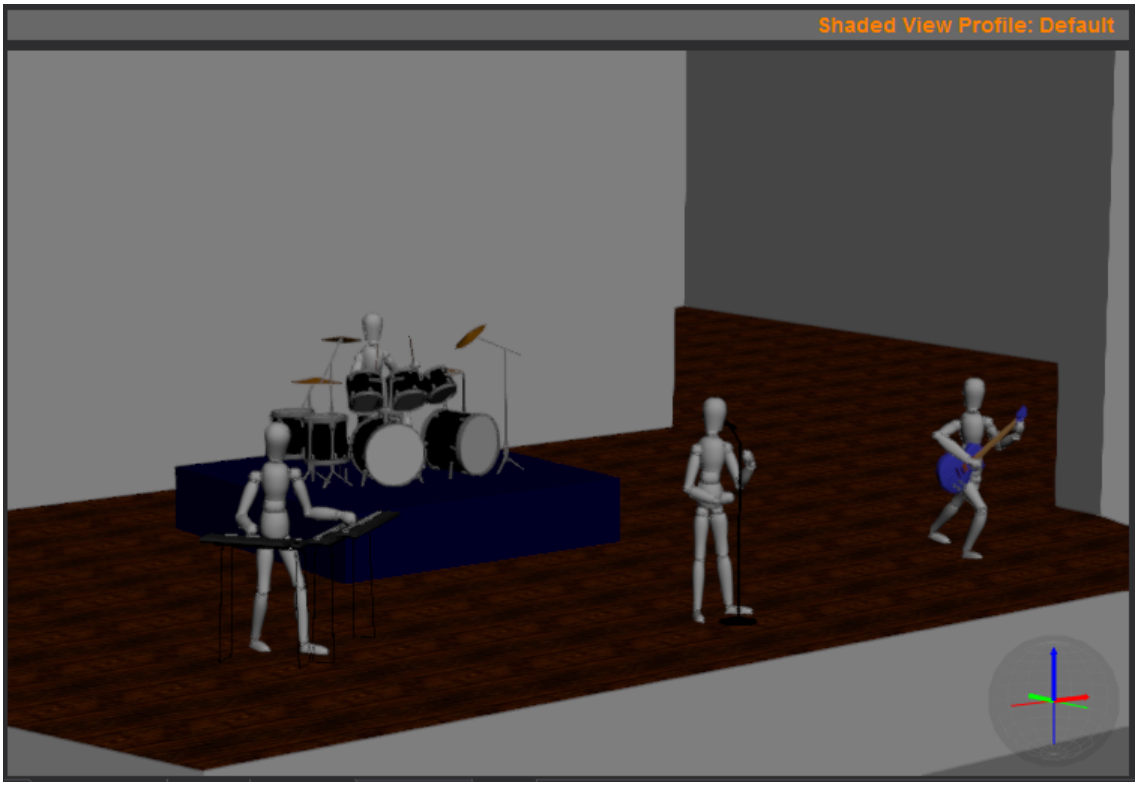


5 Scroll down to the **Wood** folder, and select **Wood 003**.

6 Click *Select*.

7 Click *OK*.

Result: The Stage Floor now appears with a brown wooden texture in the Shaded View.



Note: If you wish, you can experiment with the Stretch and Tiling properties for the texture, which will change the appearance of the texture in the Shaded View.

Enjoy customizing the rest of your Venue, Stage and Band members on your own!

Lesson 3 - Working with Hang Structures

Introduction

Once the set is in place, you can define your hang structures.

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

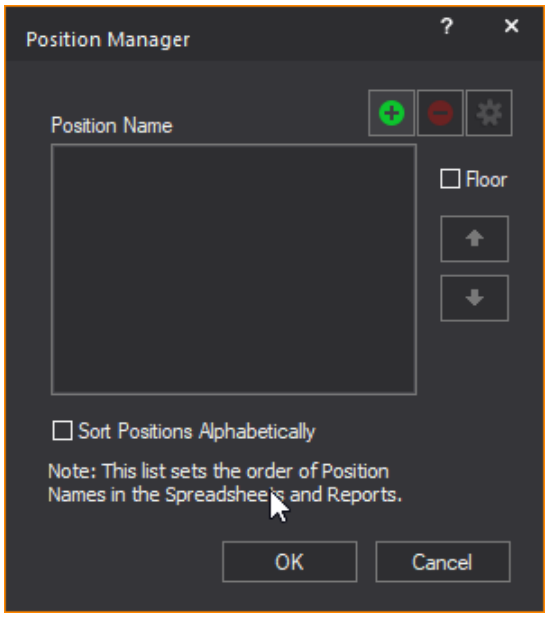
Step 1 - Defining a position name

Position names are used to identify hang structures. To organize and sort the position names, use the *Position Manager*.

Note: 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 **Managers** menu, choose **Positions**.



2 Click *New Position* button, and then enter the name of the position.

 The *New Position* button.

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

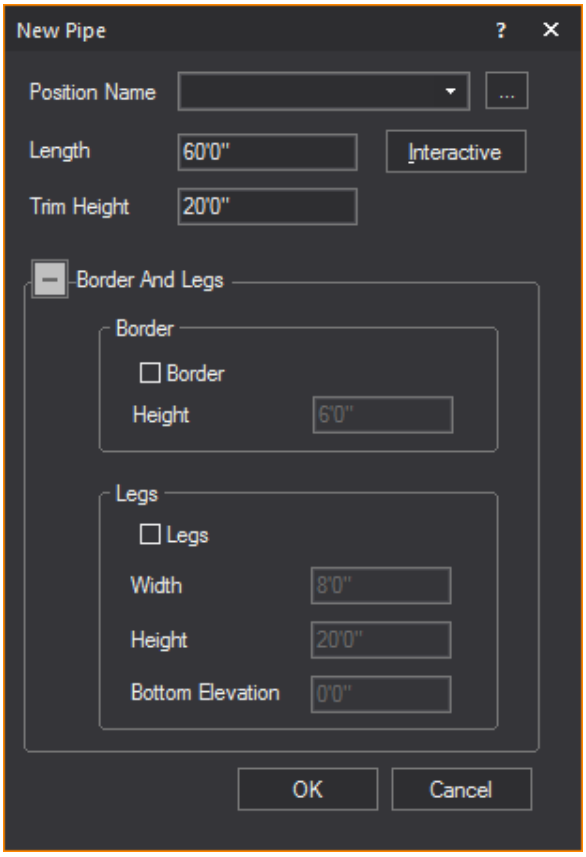
Let's start by setting our layout tab to Quad, this way we can view our stage from different viewpoints.

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

To draw a pipe

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

Result: The *New Pipe* window will open.



2 Choose the associated position name as defined in “To define a new position name” on page 29.

3 Specify the Length and Trim Height for the pipe (for this example, accept the defaults), and then click *OK*.

4 Click or type to place the pipe in your drawing. We will place this pipe above our band we created in **Lesson 2**.

Note: Since the Pipe object has a Trim Height property, this specifies its height (position along the Z axis). This is different to the objects we inserted on the stage in the previous lesson, where we set the Missing Coordinate.

Step 3 - Inserting truss

Truss hang structures can be inserted from the library.

To insert truss

1. Unlike a Pipe which has a Trim Height, Truss will be drawn at the Missing Coordinate. Since we wish to insert our Truss at **20'**, press the TAB key on your keyboard, and enter **20'** for the *CAD Coordinate* to set this as your Missing Coordinate in your Plan view
2. Ensure that Assembly Snap is enabled in the CAD Options toolbar. (It is enabled by default). If it is not enabled, please enable the *Assembly snap* tool before building your truss, so the pieces will be assembled.

    The *Assembly snap* tool button.
3. From the **Library** menu, choose **Browse Library**.
4. Click the *Truss* tab at the bottom of the *Library* browser to view the truss library

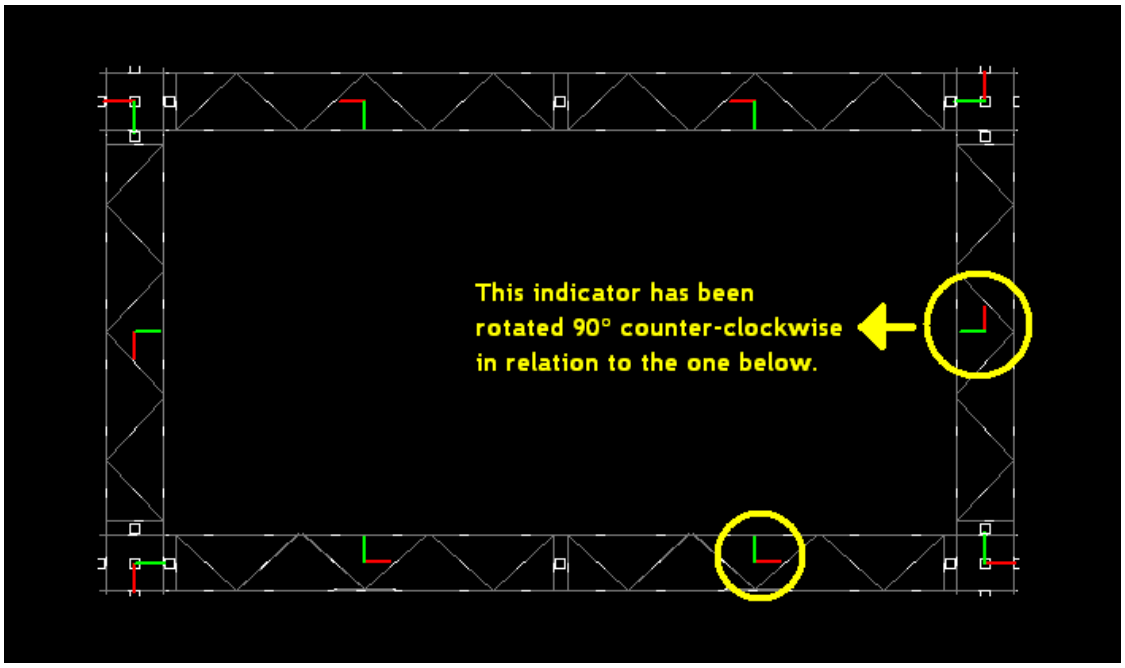
    The *Truss* tab button.
5. Navigate to, or search for, the desired piece, and then double-click in your drawing to insert it.
6. Hover your cursor over your drawing where you want to insert the object.
7. Click to place the first piece of truss.
8. To assemble subsequent pieces, hover the cursor over 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.
9. Once the piece has snapped into position, click to insert it into the drawing.
10. When you are finished placing truss, right click and select **Finish Placing Truss**.

    Note: Since you are in Quad tab, from the side or front view, note that the truss is inserted at a height of 20'.

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.



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.

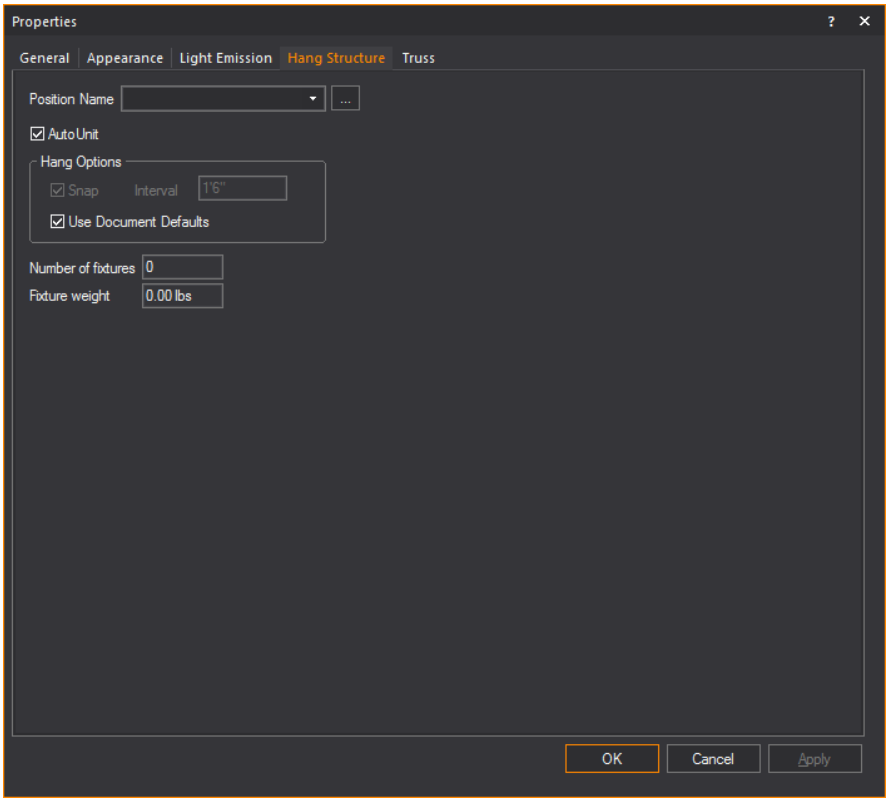
Tips:

- 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, fixtures will hang correctly from this structure.

Note: For more information on how to assemble truss, please refer to the "Hanging Truss" section in the Reference Guide.

To assign a position name

1 Double-click the truss structure to open its *Properties* window.



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

3 Click *OK*.

To insert Truss Vertically

Truss can be inserted on the Floor standing on one of its ends.

1 From the *Library Browser*, in the *Truss* tab, locate a straight section of Truss, e.g. **Christie** > **Type A** > **12x12 8ft**.

2 Right-click on the **Truss name** in the *Library Browser* and select one of the **Insert Vertical** options. The rotation angle selected will determine the direction that the truss' Ladder side will face.

3 Move your cursor over the drawing area, and click to insert the Fixture.

Note: When Truss is inserted vertically in Plan View, the Missing Coordinate defines the height the truss’ base is inserted.

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:

- Insert fixtures
- Focus fixtures
- Relocate and Clone fixtures

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

Step 1 - Inserting fixtures

Fixtures are objects on the plot which can be hung on a hang structure, or inserted on the floor.

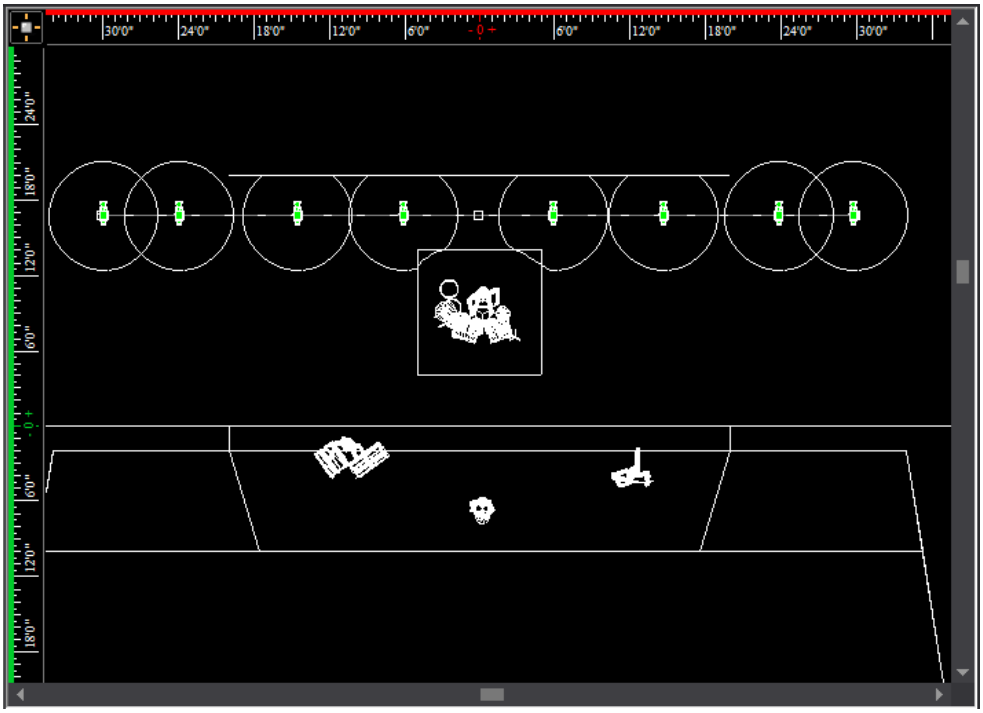
To hang fixtures

1. In **CAD** mode in Wireframe view, from the **Library** menu, choose **Browse Library**.
2. Select the *Fixtures* tab at the bottom of the *Library Browser*, to ensure that you are browsing the fixture library.

    The *Fixtures* tab button.
3. Navigate to the desired item, right-click, and then select **Insert**. For this example we will select **Source 4.**
4. Move the mouse over a hanging structure (i.e. pipe or truss) and click to hang the fixture. We will hang the **Source 4** on the pipes created previously in lesson 3.

5 Place 8 **Source 4** on the pipe. When you have finished placing the fixtures, right click and select **Finish Placing Fixtures.**

Result: The **Source 4** fixtures will be inserted onto the pipe.



Tip: Whenever you hang fixtures on a truss, it is always best to work in *Isometric View* and *Zoom In* to make precise fixture placement on specific truss chords.

Inserting Fixtures on the Floor

Automated Fixtures can be inserted on the floor, which does not require using a Pipe or Truss.

1 From the *Library Browser*, in the *Fixtures* tab, locate an Automated Fixture, such as a **Clay Paky Sharpy**.

2 In the *Library Browser* right-click on the Fixture name and select **Insert on Floor**.

3 Move your cursor over the drawing area, and click to insert the Fixture.

Note: When inserting Fixtures on floor, they will be placed at the specified Missing Coordinate height.

Step 2 - Focusing fixtures

There are three methods for focusing fixtures:

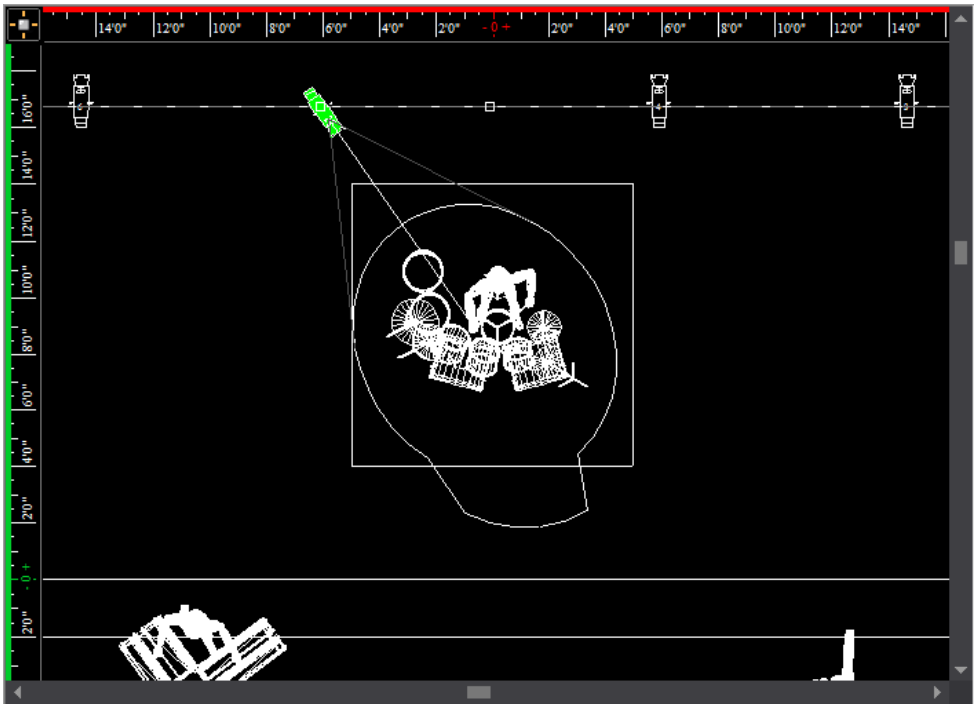
- clicking and dragging the light beam of the selected fixture
- changing the pan and tilt values by double-clicking to access the fixture's properties box

- inserting focus objects, such as a Focus Position, a Focus Line or a Focus Arc.

To manually drag a fixture's light beam

1 In **CAD** mode in Wireframe view, click on a fixture.

Result: The fixture will be selected. The fixtures beam will appear.

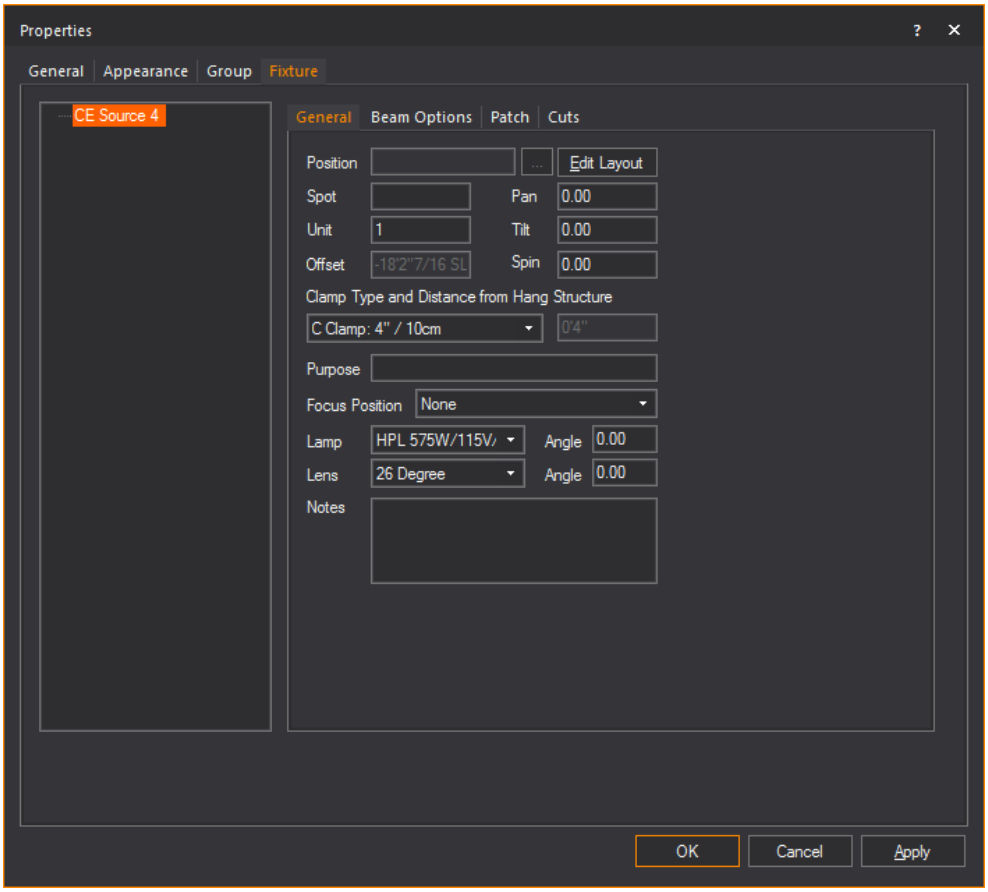


2 Click and drag the beam to where you want it positioned.

To focus a fixture using its properties

1 In **CAD** mode in Wireframe view, double click on a fixture.

Result: The *Properties* window of the fixture will appear.



2 In the *Fixture* tab, in the *General* subtab, look *Pan* and *Tilt* fields. These values correspond to the focus drag of the fixture. By changing these values we change the position of the beam.

3 In the *Pan* field, enter in **30**.

4 In the *Tilt* field, enter in **45**.

5 Click *Apply*.

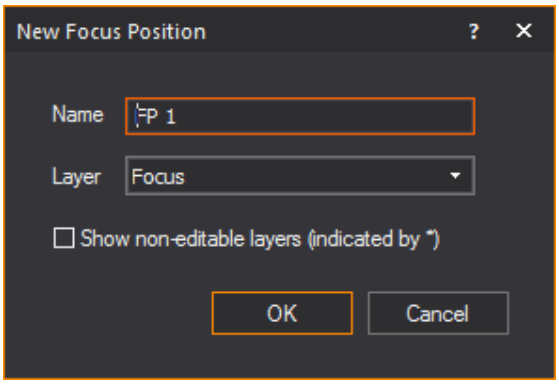
Result: The fixture will move according to the values entered changing the beams location.

Focus Objects allow you to focus one or more fixtures to a specific Focus Point in space (i.e. coordinate at which the Focus Position was inserted). Or, it will allow the focus of multiple fixtures to be evenly spread along a Focus Line or Focus Arc.

To insert a Focus Position

1 From the **Draw** menu, choose **Focus Position**.

2 The *New Focus Position* window will open.



3 In the *Name* field, type a label for the new focus position and then click *OK*. For this example call it "**Lead Singer**"

4 Press TAB to adjust the Missing Coordinate, if necessary.

5 Click to place the focus position in the drawing. Place this one over the **Person with Microphone** created in **Lesson 2**.

6 Create additional focus positions for the **Person with Electric Guitar, Person at Keyboard** and **Person at Drum Kit.**

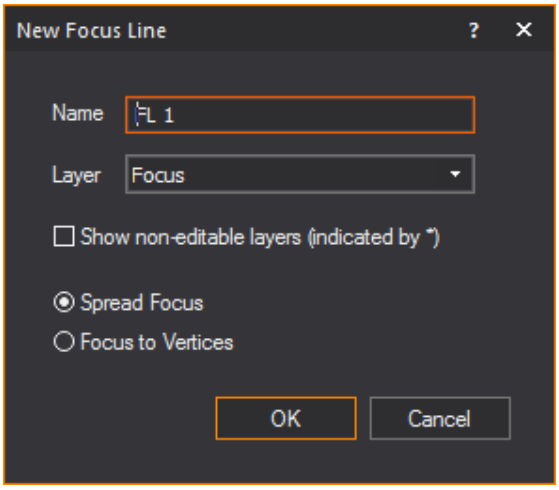
Focus Lines and Focus Arcs

Focus Line and Focus Arc are special focus objects which multiple conventional fixtures can be assigned to, and spread out evenly across the object, or focused to the vertices (in the case of a Focus Line).

To insert a Focus Line

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

Result: The *New Focus Line* window appears.



2 In the *Name* field type a name for the new Focus Line.

3 In the *Layer* drop-down menu select the layer on which you want to draw the Focus Line.

4 Select the focus type.

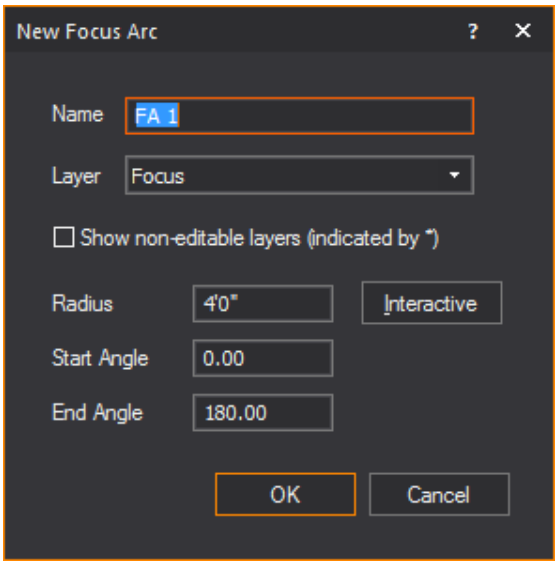
5 Click *OK*.

6 Draw your Focus Line as required.

To insert a Focus Arc

1 From the **Draw** menu, choose **Focus Arc**.

Result: The *New Focus Arc* window appears.



2 In the *Name* field type a name for the new Focus Arc.

3 In the *Layer* drop-down menu select the layer on which you want to draw the Focus Arc.

4 To drawn the focus arc free hand, click *Interactive*. To pre-construct the focus arc, enter in the radius, start angle and end angle details, then click *OK*.

5 Place your Focus Arc as required.

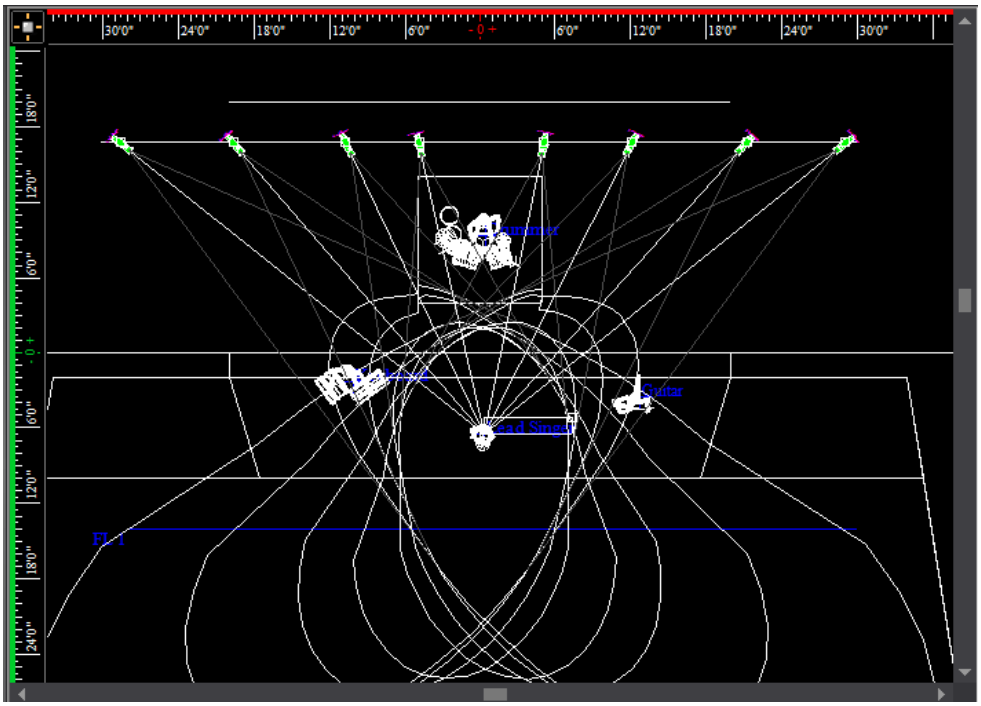
Step 3 - Focusing fixtures using focus positions

Note: Focus Objects will also work with Automated Fixtures, and can be used with the Focus Designer tool. This will be covered in a Lesson 6 - Building lighting Looks in Design mode.

To assign fixtures to a Focus Position

1. In **CAD** mode in Wireframe view, from the **Tools** menu, choose **Quick Tools** > **Quick Focus**.
2. Click on the focus position to set it as the active focus position. For this example, select the **Lead Singer** Focus Position
3. Click on the desired fixture(s) to set there focus to the selected Focus Position. In this example click all the **Source 4** fixtures.
4. When you are finished focusing fixtures, right-click and select **Finish Quick Focus**.

   Result: All the fixtures will be focused on the **Singer** Focus Position.

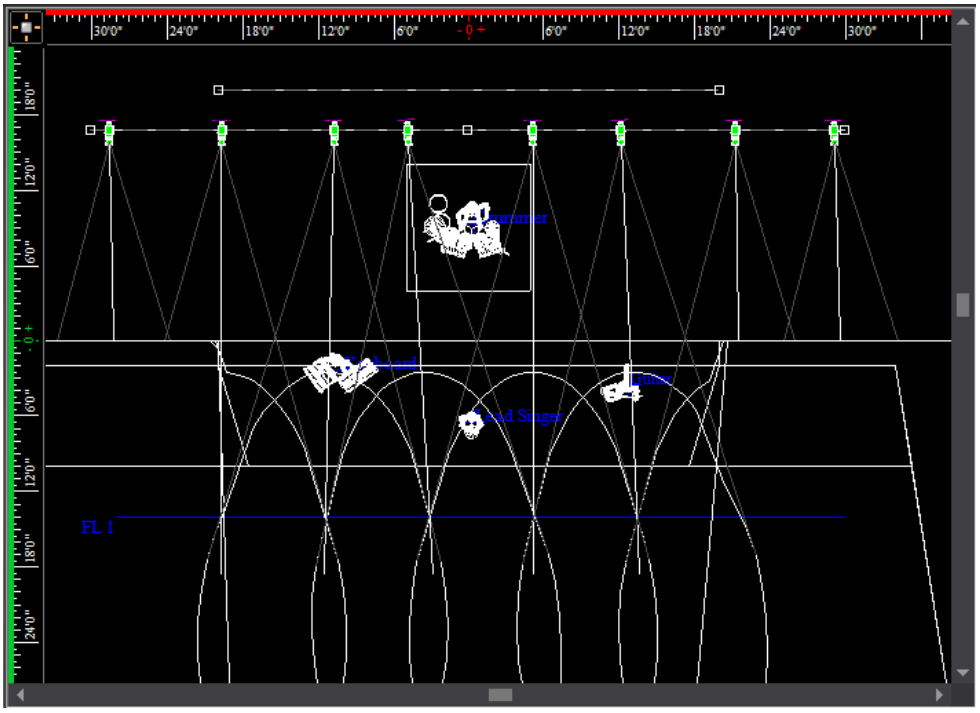


To assign fixtures to a Focus Line

1. In **CAD** mode in Wireframe view, from the **Tools** menu, choose **Quick Tools** > **Quick Focus**.
2. Select the fixtures you want to focus and also select the Focus Line to which you want to assign conventional fixtures in CAD mode.

3 When you are finished selecting, right-click and select **Finish Quick Focus**.

Result: The fixtures will be focused along the Focus Line



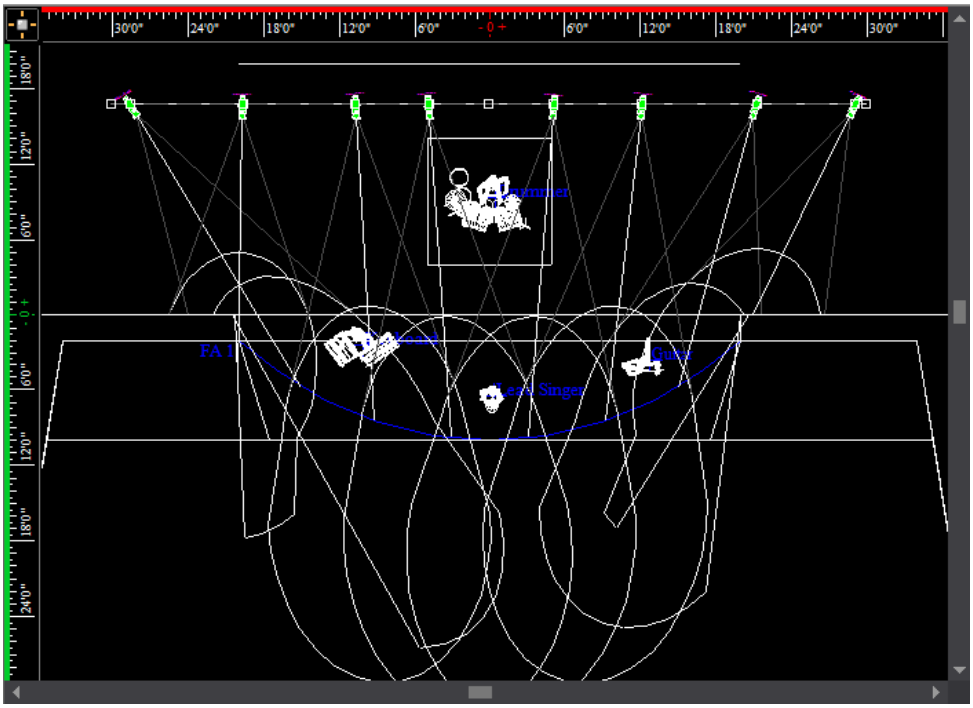
To assign fixtures to a Focus Arc

1 In **CAD** mode in Wireframe view, from the **Tools** menu, choose **Quick Tools**, **Quick Focus**.

2 Select the fixtures you want to focus and also select the Focus Arc to which you want to assign conventional fixtures in CAD mode.

3 When you are finished selecting, right-click and select **Finish Quick Focus**.

Result: The fixtures will be focused along the Focus Arc.



Step 4 - Relocating or cloning fixtures

Move or copy multiple fixtures at once from one hang structure to any other hang structure in Wireframe. The steps to use **Relocate Fixtures** and **Clone Fixtures** are the same.

Tip: Whenever you relocate or clone fixtures from one truss to another truss, it is always best to work in Isometric View and Zoom In to make precise fixture placement on truss chords.

To relocate or clone fixtures

1 In CAD mode in Wireframe view, select two or more fixtures on a hang structure.

2 From the **Edit** menu, choose **Relocate Fixture** to move fixtures or **Clone Fixtures** to copy fixtures.

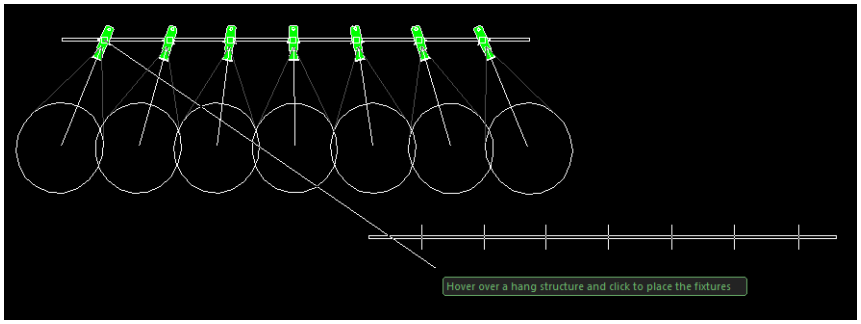
Tip: You can right-click on the selected fixtures, then choose **Relocate Fixtures** or **Clone Fixtures** from the menu that appears, or use the corresponding keyboard shortcuts CTRL+X or CTRL+C.

Note: For these functions to work, only select fixtures on a single hang structure or the same chord of a truss.

3 Hover over the target hang structure to where you want to relocate or place the copied fixtures.

Result:

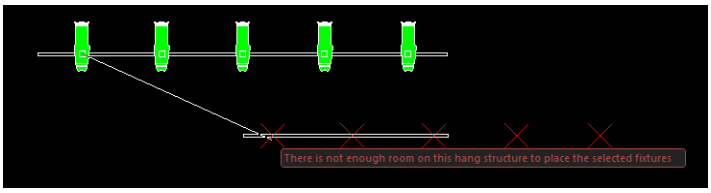
Line markers appear on the target hang structure to indicate where the fixtures will be placed after the move or copy is completed.

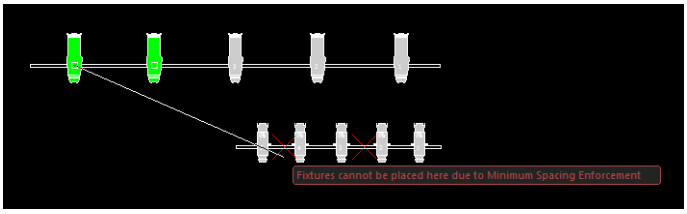


OR

Red *X markers* appear if:

- The target hang structure does not have enough space for the selected fixture.
- The *Enforce Minimum Spacing* value set in *Document Options* > *Hang Structure Settings* is considered when placement is validated.





4 (This step is optional.) With fixtures on the cursor, right-click to select and enable any of the functions from the menu that appears.

- *Reverse Direction*: Relocated or cloned fixtures are placed in the opposite direction from their original locations.
- *Flip Pan*: The Pan angles of the relocated or cloned fixtures are flipped by 180°.

  Example: When a fixture with Pan angle set to -30° is relocated, its Pan angle will change to 150°.
- *Hang Fixtures on Hang Position 1*: Relocated or cloned fixtures are placed on the bottom truss chord of the same truss.

- *Hang Fixtures on Hang Position 2*: Relocated or cloned fixtures are placed on the top truss chord of the same truss.

Notes:

- The functions remain enabled until they are disabled.
- *Reverse Direction* and *Flip Pan* can fix the focus and orientation of fixtures if reversed after relocating and cloning.
- *Reverse Direction* and *Flip Pan* can be used at the same time.

5 (This step is optional.) With the fixtures on the cursor, right-click and select *Remove Resulting Fixtures' Attributes* from the menu that appears. Selecting this option will clear the fixture(s) of attributes after clone or relocation.

Notes:

- The *Unit* number will be assigned to the cloned or relocated fixture(s) automatically if the target hang structure is a Pipe and the *AutoUnit* checkbox in the *Hang Structure Settings* tab of *Document Options* is selected.
- If you select *Remove Resulting Fixtures' Attributes* menu, *Mode*, *Lens*, and *Lamp* attributes will not clear.
- If you selected *Remove Resulting Fixtures' Attributes* menu during a previous Relocate/Clone operation, it is not selected for the next operation.

6 Click to drop and place the fixtures on the target hang structure.

Result: The selected fixtures are moved or copied to the target hang structure.

Notes:

- Conventional fixtures maintain their original focus and orientation after you relocate or clone.
- You can relocate or clone to and from different types of hang structures if they fit the target space.
- You can relocate imported fixtures from collinear pipes to other hang structures.

Lesson 5 - Assigning fixture attributes

Introduction

In this lesson you will learn how to assign control data and physical details of fixtures, including colors and gobos. You will also learn how to change and otherwise manipulate this data, which is referred to as "Fixture Attributes".

Step 1 - Assigning attributes to fixtures

Once you have determined what fixtures you wish to use for your show and where to place them, you will likely need to define how they are to be used.

> **Example:** What circuit will a conventional fixture be plugged into? What dimmer will that circuit be connected to? What channel on your console will control the intensity of this dimmer/fixture? Will this fixture require a gel, a gobo, both, or some other accessories such as a set of barndoors, a tophat, a scroller, etc.? How will your automated and LED fixtures be patched?

Once you have made these decisions, you may assign the necessary Fixture Attributes in one of three ways:

- By using *Quick Tools*.
- By accessing the fixture's properties and entering data into the fields found on the various sub-tabs under the *Fixture* tab.
- By entering the information into one of the spreadsheets.

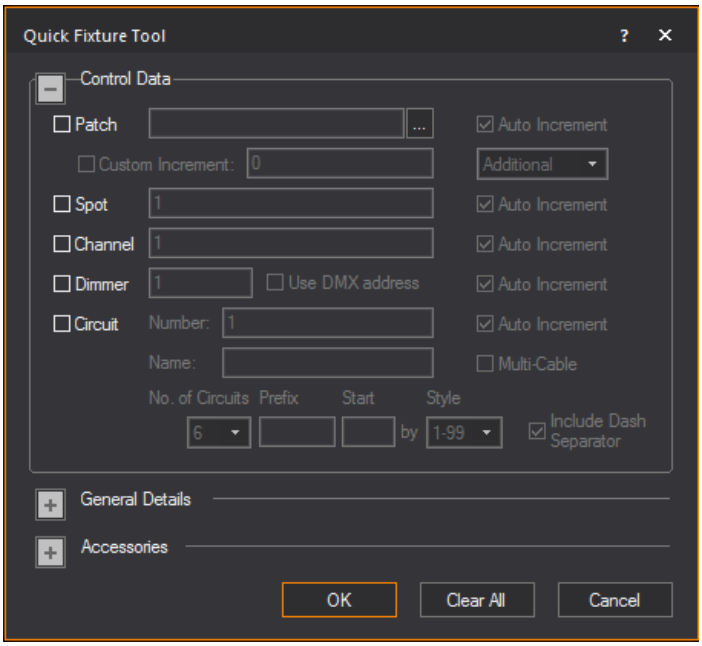
Quick Tools

To use Quick Tools to assign attributes to fixtures

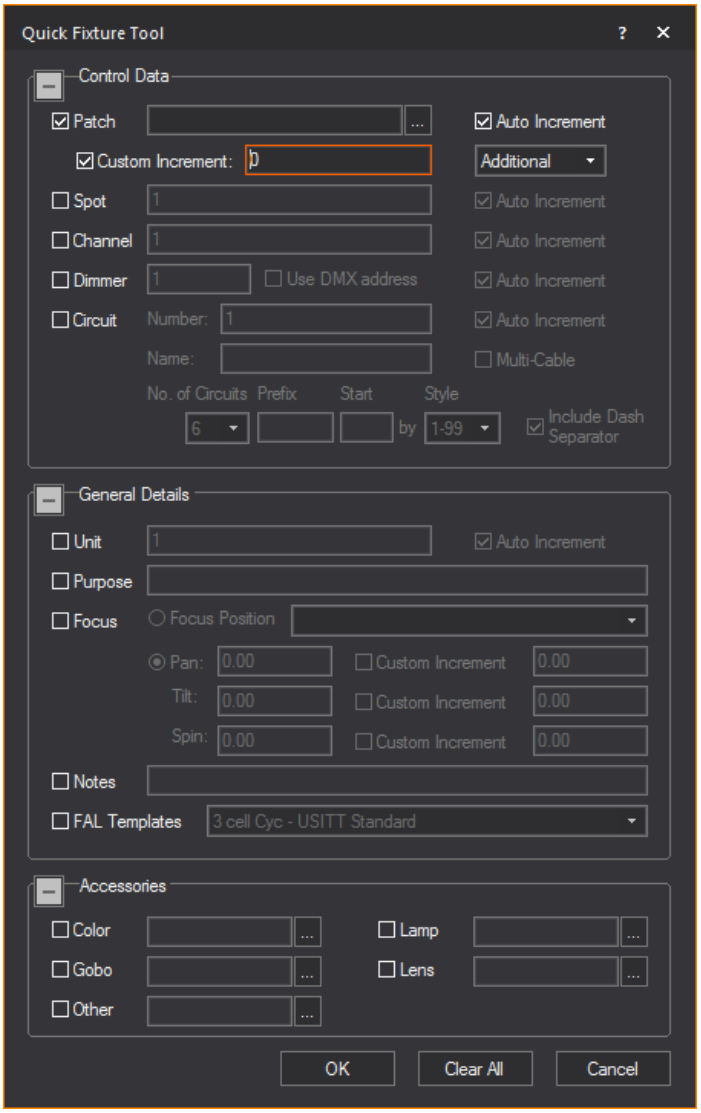
1 Working in the Wireframe view, go to the **Tools** menu, **Quick Tools** > **Quick Tools** launch Quick Tools.

Note: *Quick Tools* can also be opened by pressing the Q key on your keyboard, or clicking the *Q* button on the *Tools* toolbar.

Result: The *Quick Fixture Tool* window will open.



2 Click the + button beside each heading (Control Data, General Details, Accessories) to access the attributes they contain.



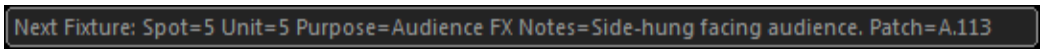
3 Mark the checkbox next to the attribute(s) you wish to assign.

4 Type or select the details required for each attribute and mark the checkboxes for any required incrementing options.

Note: You may choose to apply only one or any number of attributes at once. In order to avoid potential problems though, it is recommended that you do not assign more than a handful of attributes at the same time.

5 Click *OK* to start the Quick Tools session.

Result: The *Quick Fixture Tool* window will close. A "**Q**" appears near the cursor. If Instruction Tooltips are enabled (**Options** >**Application Options,** in the *General* tab) the tooltip near the cursor shows what details are going to be assigned to the next fixture you click on. The same information can also be found on Status bar at the Prompt Line.



Note: As you hover over a fixture in the wireframe, the fixture will be highlighted orange. This is to indicate the attribute(s) you have just defined will be assigned to that specific fixture once you click on it.

6 Click on the required fixture to assign the desired attribute(s) to it.

Result: If sound is enabled on your computer, a "**Beep**" will sound to confirm that the selected attribute(s) have been applied. If any incrementing options were selected, the Instruction Tooltip and Status bar will update to show the new values that are going to be applied to the next fixture you click on.

7 Repeat **Step 6** for all fixtures you wish assign attributes to.

8 Once you're done assigning attributes for this session, press the Enter or Esc key to complete the session.

Note: The session can also be completed by right-clicking and selecting **Finish Quick Tools**.

Note:

- After using *Quick Tools* to assign attributes to a fixture, you may click on the same fixture again, during another Quick Tools session. The current Quick Tools session will override the previous.
- Patch control data must fulfill the requirements of patch notation, which is "*[universe_name].[DMX_address]".*
- After clicking to assign attributes to a multi-circuit or multi-patch fixture, a dialog appears, listing the circuit names to which you can assign the selected Attributes. You may select any number of circuits, then click *Select* to actually assign the attribute(s).
- When using *Quick Tools* to insert a color or gobo, WYSIWYG automatically inserts the default color frame or gobo holder defined for that fixture.

Quick Tools increment feature

The *Auto Increment* and *Custom Increment* options found in the *Quick Fixture Tool* window can be enabled in order to assign sequential data to fixtures.

Example: If you enable *Auto Increment* for the Unit and Dimmer attributes, *Quick Tools* will automatically assign the next logical number to the next fixture you click on, starting with the value that entered in the *Quick Tools* window. One of Auto Increment's most useful purposes is assigning Patch data.

Using the Quick Tool Increment feature

1. In Wireframe view, press *Q* to open the *Quick Tools* window.
2. Expand the Fixture Data section and mark the *Patch* checkbox.
3. In the *Patch* field, enter the name of the DMX universe to which you wish to patch the fixture, followed by a period (**.**) and then by the starting DMX address.

   Attention: Ensure that there are no spaces anywhere in this field.
4. To the right of the *Patch* field, mark the *Auto Increment* checkbox.
5. Click *OK*.
6. Click on the first fixture in the patch sequence.

   Result: The patch value is assigned to the fixture and the Tooltip and Status bar change to show the next patch value. If, for example, the fixture you just clicked on requires 17 DMX channels, and you started the patch at A.1 (i.e. Universe "A".DMX Address "1") the patch value that now appears in the Instruction Tooltip is A.18. This patch value will be assigned to the next fixture you click on.

The *Auto Increment* option works the same for all attributes where it can be enabled. When enabled, values will increase by increments of 1. Please consult the WYSIWYG Reference Guide for information on how to use the Multi-Cable option for Circuits.

Using the Quick Tool Custom Increments

Custom Increments are used for assigning sequential Pan, Tilt or Spin angles to fixtures.

Attention: In this example, you will focus Par fixtures on a bar of six panned 10º apart from each other but tilted at the same 45º angle.

1. In the *Quick Fixture Tool* window, in the Fixture Attributes section mark the *Focus* checkbox.
2. Mark to select the *Pan/Tilt/Spin* radio button.
3. To the right of the *Pan* value field, mark to check the *Custom Increment* checkbox .
4. In the *Custom Increment* field for Pan, enter **10**.
5. In the *Tilt* field, enter **45**.
6. Click *OK*.

   Result: As you click on each fixture on your bar of six, their Pan and Tilt will change to the values displayed in the Instruction Tooltip before you clicked.

Changing Quick Tools options mid-session

It is possible to change any of the attribute details while within a Quick Tools session.

> **Example:** If you wish to assign the same Dimmer number to the next conventional fixture, but you've enabled Auto Increment for Dimmers, here is how:

1. Press the Q key on your keyboard, or right click and select **Change Quick Tools Options**.

   Result: The *Quick Fixture Tool* window will open.

2. In the *Quick Fixture Tool* window, make the necessary changes to the attributes.

3. Click *OK* to resume the session.

Fixture Properties

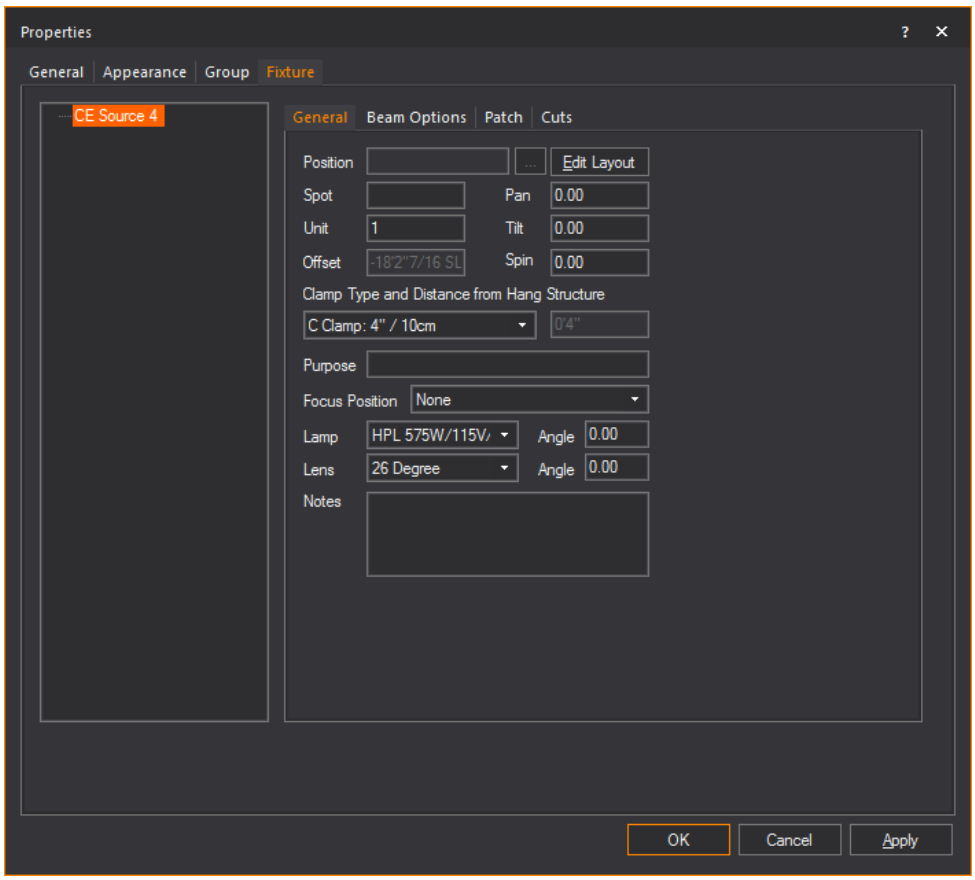
Another way to assign attributes to fixtures is by their *Properties* window. Since this method involves quite a lot of clicking, it is best used for fixtures that are in some way unique (i.e. they do not follow any of the "patterns" that can be defined or achieved by using *Quick Tools)* and/or to view, update or modify Attributes that have already been assigned. To use a fixture's *Properties* window:

To assign attributes to fixture by the Properties window

1. In Wireframe view, double-click on a fixture which you have assigned one or more attributes.

   Result: The *Properties* window opens.

2 In the *Properties* window, click the *Fixture* tab.



3 Various attributes of the fixture can be found under the sub-tabs *General*, *Beam Options*, *Patch* and *Cuts*. These attributes can be modified as desired.

4 To apply the changes and keep the *Properties* window open, click *Apply*. To close the *Properties* window and apply changes, click *OK*.

Attention: Clicking *Cancel* will close the *Properties* window without applying any changes, except for changes made before the *Apply* button was last clicked.

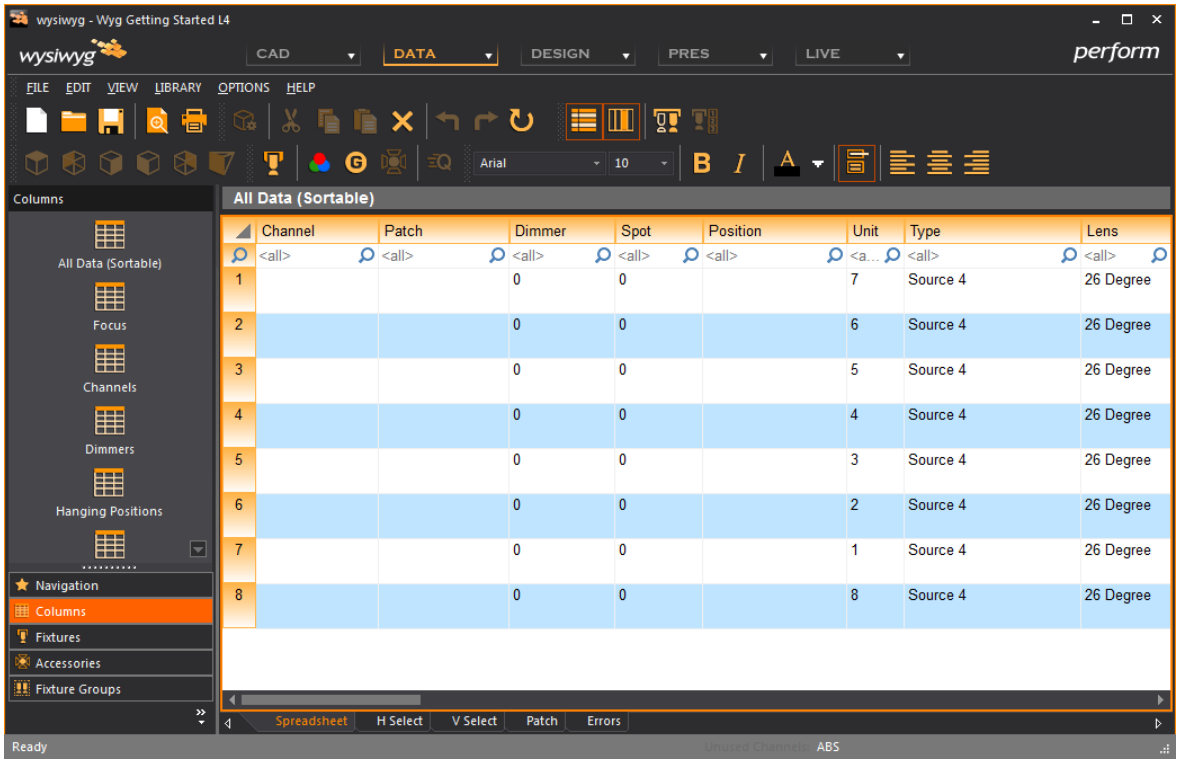
In order to avoid mistakes, it is best to open the *Properties* window of fixture for a single fixture at a time.

However, should you wish to access the properties for multiple fixtures at once, simply select the fixtures, right-click and then select **Properties** or press ALT+Enter. If multiple fixtures (regardless of type) are selected, modifying one or more attributes via the *Properties* window results in *all* selected fixtures having the new values applied to them.

Spreadsheets

As you place fixtures into your file in the CAD wireframe, WYSIWYG automatically adds them to its internal spreadsheet. The spreadsheet is always synchronized with the wireframe, so any changes made in one will be reflected in the other immediately. In other words, if you were to delete a fixture from the Spreadsheet view, it would disappear from the Wireframe view as well—and vice-versa.

To access the spreadsheet, simply click a view tab which contains a spreadsheet (such as the *Spreadsheet* tab in DATA mode).



The Spreadsheet view contains not only the full inventory of all the fixtures in your file, but all of their attributes' values, as well as other information—some of which is only available in spreadsheets, such as fixtures' absolute position in 3D space or the type of Circuit they require. Each row in the spreadsheet represents a single fixture, or a single cell/circuit of a multi-cell/-circuit fixture, or a single electronic accessory associated with a fixture, while each column contains the values of fixture attributes.

A Fixtures' visibility in the spreadsheet is controlled by layers' visibility (i.e. if a layer is not visible, then any fixtures that reside on that layer will not be visible either). The visibility is also controlled by the **Show Selected Only** option (i.e. when this option is enabled on the Spreadsheet toolbar or in the **View Options** of the spreadsheet, only fixtures selected in a wireframe will appear in the spreadsheet). Please note that layers supercede the **Show Selected Only** option: when a layer is turned off, it is impossible to select anything it contains. Attributes' (i.e. columns) visibility is controlled from the spreadsheet's **View Options**, as follows:

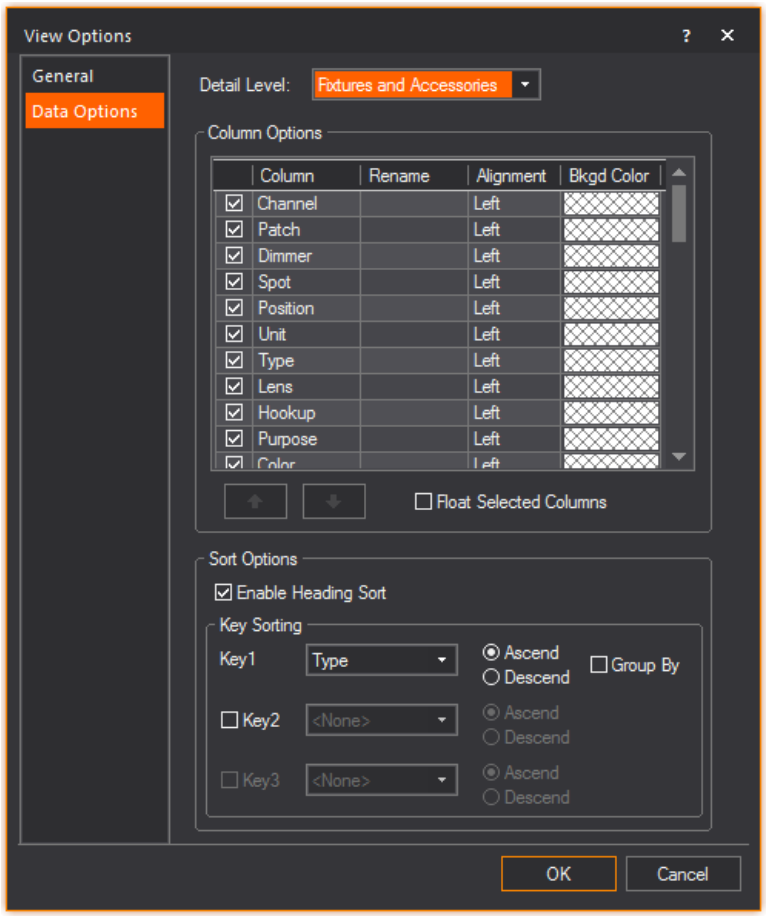
To modify the columns of a spreadsheet

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

Result: The *View Options* window will appear.

Note: You can also right-click on the spreadsheet shortcut and select **Properties**.

2 Click the *Data Options* tab.



3 To remove a column from the spreadsheet, under the Column Options heading, unmark the checkbox next to that column's name.

4 For the changes to take effect, click *OK*.

To modify fixture attributes directly in the spreadsheet

1 Click the cell that represents the fixture and attribute you wish to modify.

2 Type the new value or select it.

3 Press Enter on the keyboard.

Result: The value is assigned to the fixture attribute.

It is also possible to assign the same value to multiple attributes at the same time. For example, if you wish to assign the same colour to multiple conventional fixtures:

To modify the attributes of multiple fixtures simultaneously

1. Select the cells under the column you want to modify. For this example, select all the cell in the Color column.
2. Enter in the new value. For this example, type R57 (for Rosco 57 “Lavender”).
3. Press Enter on the keyboard.

   Result: The value is assigned to the fixture attribute. A Rosco 57 gel is added to each fixture whose Color cell you selected.

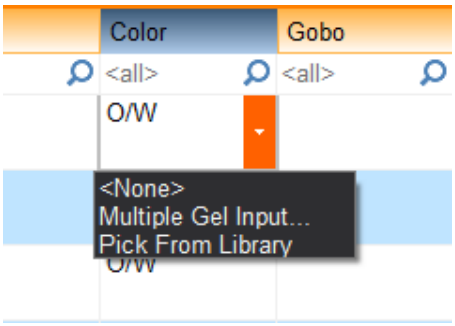
Step 2 - Choosing data

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.



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 - 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.

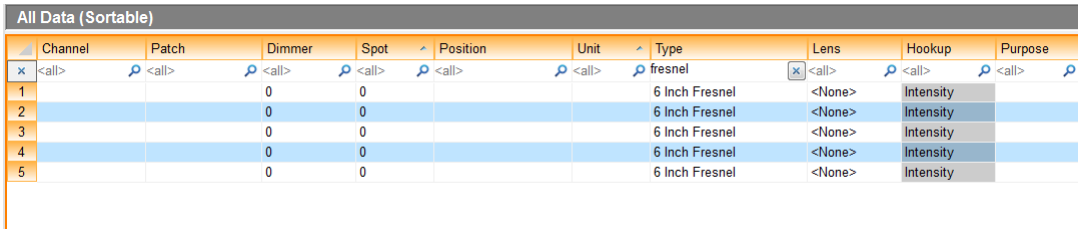
Begin in the spreadsheet. You can use data filters to quickly locate data in the spreadsheet.

To use a data filter

1 In the Spreadsheet view, 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 and then hit ENTER.

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.

Lesson 6 - Building lighting Looks in Design mode

Introduction

Design mode offers a creative environment for lighting designers to use the design tools to experiment with lighting, video and automation to create cue concepts. You can create static looks, export them as images or render them for design presentations.

Design mode is intended so you do not need channel numbers or a patch. You can 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.

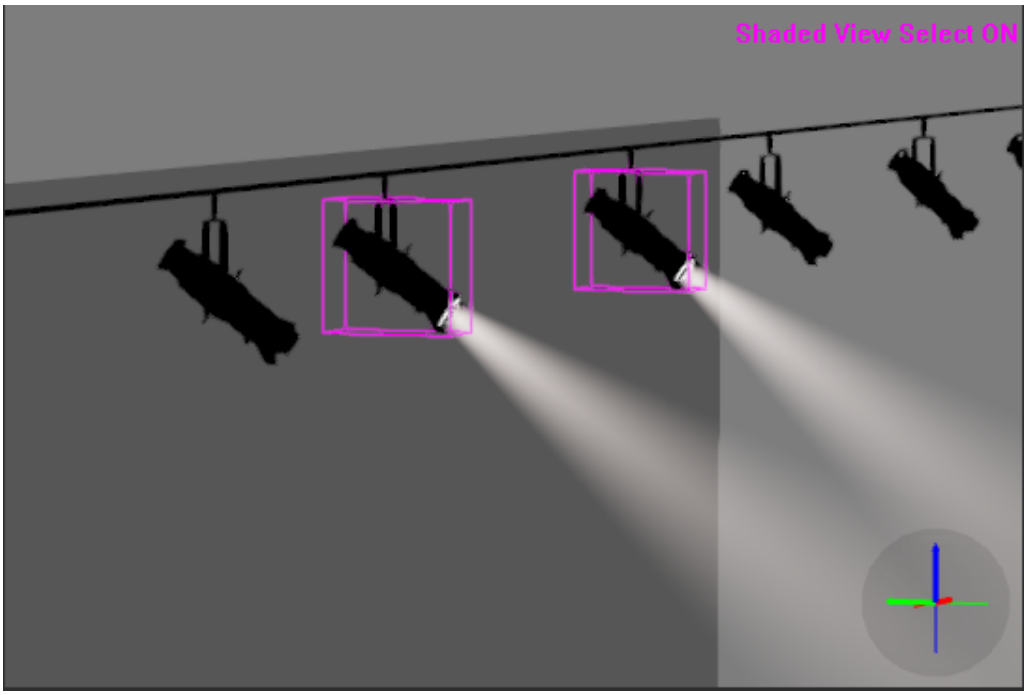
Step 1 - Fixture selection in Shaded view

To create a Look, you will have to select and manipulate fixtures in Shaded view. A fixture can be selected from the Shaded view similar to how fixtures can be selected in Wireframe view. This step will teach you how.

To activate Shaded View Selection

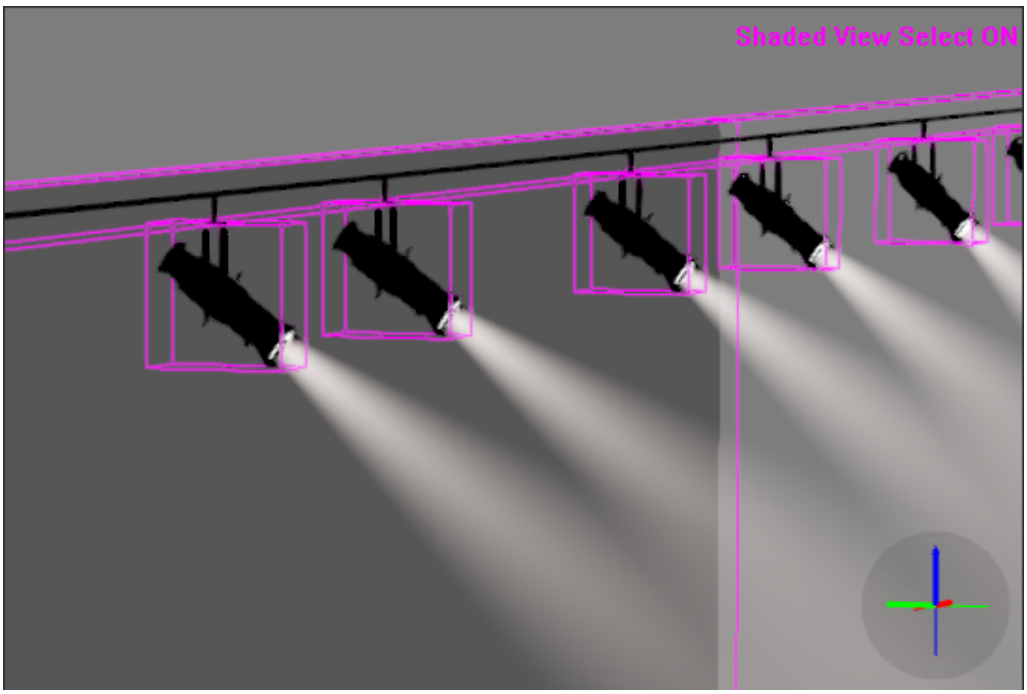
1 In a Shaded view in CAD, Live or Design mode (meaning the amber outline appears around the Shaded view), press *TAB* once. You are now in Shaded View Selection mode.

2 To select a fixture in Shaded view Selection mode, click directly on a fixture's image in the Shaded view:



3 To select multiple fixtures in Shaded View Selection, press CTRL and click **each fixture that you want to select**.

You can also select multiple fixtures at once by drag-selecting with the mouse as you would in wireframe modes; Shaded View Selection disables the camera movement controls so you can drag a marquee box around fixtures to select them.



4 Once you are finished selecting fixtures, press TAB again to exit from Shaded View Selection mode. You will then regain control of the Shaded View camera.

Tip:

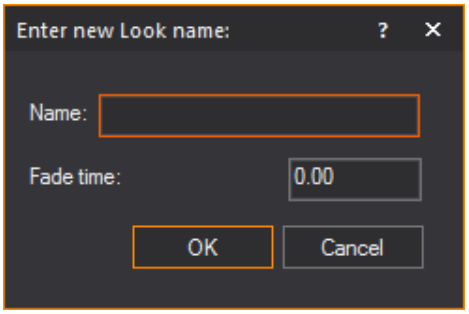
- It is helpful to keep your hand near the *TAB* key while selecting fixtures and maneuvering around with the camera; once you need to move around your view, press *TAB* to exit Shaded View Selection. Your fixture selection is maintained as you move in and out of the Shaded View Selection mode.
- Make sure while you navigate around in the Shaded view that you give yourself a good angle on the view in which you want to select fixtures. It may be difficult to select a fixture in the middle of a lighting position or truss while you're looking at a side view of it.

Step 2 - Creating a new lighting Look

To create a new Look

1 In the Looks shortcut bar area, right-click and select **New Look**.

Result: The *Enter new Look name* window will open.



2 In the *Enter new Look name* window, 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 3 - Using the Intensity tool

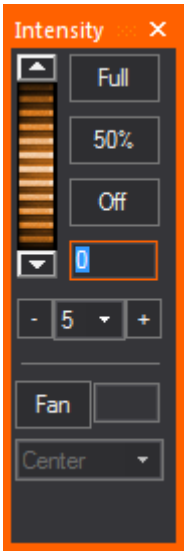
To use the intensity tools

1 In your drawing, select the desired fixtures.

2 On the Design toolbar, click the *Intensity Designer Tool* button.

The *Toggle Intensity Designer Tool* button.

Result: The *Intensity* window will open.



3 Click the dial, and then move the mouse up or down to set the intensity level, or use the up/down arrows for single increment/decrements.

4 Click the buttons for setting intensity to *Full* (100%), *Half* (50%) and *Off* (0%).

5 You can set increment/ decrements by 5%, 10% or 25% and adjust the intensity by using the + (plus) or — (minus) buttons.

Note: The Zoom and Iris tools work the same way.

Step 4 - Using the Focus tool

To use the focus tool

1 On the Design toolbar, click the *Focus Designer Tool* button.

The *Toggle Focus Designer Tool* button.

Result: The *Focus* window will open.

2 To focus automated fixtures, with the fixtures selected, choose from any of the following methods:

- Use the *Home* button to focus the fixture back to its home position, which resets the fixture to pan = 50% and tilt = 50%.
- 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 from the *Focus Objects* drop-down list.
- Use the *Focus Pad* by clicking on the pad, and then moving the mouse.
- Use the *Pan* wheel and *Tilt* wheel to control pan and tilt of the fixture.

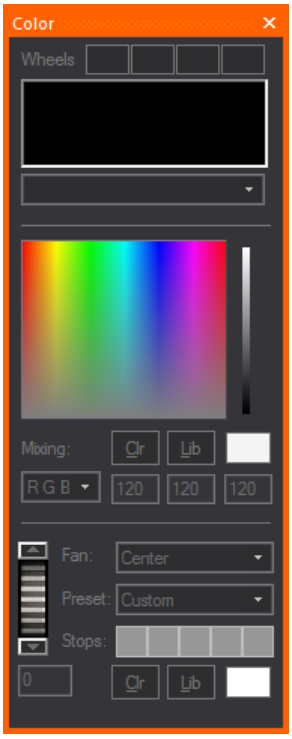
Step 5 - Using the Color tool

To use the color tool

1 On the Design toolbar, click the *Color Designer Tool* button.

The *Toggle Color Designer Tool* button.

Result: The *Color* window will open.



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 - Using the Gobo tool

To use the gobo tool

1 On the Design toolbar, click the *Gobo Designer Tool* button.

 The *Toggle Gobo Designer Tool* button.

Result: The *Gobo* window will open.

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.

Note: The Prism toolbar work the same way.

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.

 The *Fade Looks* button.

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**.

Step 8 - Using CueLists

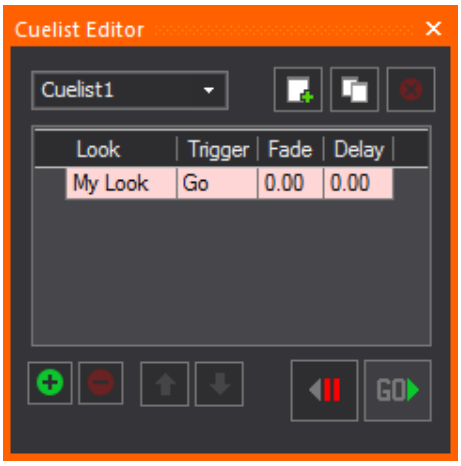
Cuelists enable you to play back stored Looks in any order, without having to manually start the crossfade function described in the previous section.

To create a Cuelist

1 On the Design toolbar, click the *Cuelist Editor Tool* button to open the Cuelist tool.

The *Toggle Cuelist Editor Tool* button.

Result: The *Cuelist Editor* window appears.



2 The *Cuelist Editor* window, click the *Create a new Cuelist* button.

The *Create a new Cuelist* button.

3 In the window that appears, type a name for the Cuelist, and then click *OK*.

To add a Look to a Cuelist

1 Once you add a cue, you can assign a Look to it by clicking the drop-down in the *Look* column and selecting the appropriate Look.

2 In the *Trigger* column, set the trigger to **Follow** or **Go**

- **Go -** The fade for the current cue will start immediately (the delay time is 0 seconds).
- **Follow -** The fade for the current cue will activate automatically after the set delay time has passed.

3 In the *Fade* column, set the Fade time. A Fade is the time (in seconds) that it takes to transition between the previous cue and the current cue.

4 In the *Delay* column, set the delay time. A delay is the time (in seconds) that it takes for lights to start fading into the current cue.

5 Once a few Looks have been added in the desired order, and the fade/delay times have been set, click on the *GO* button to watch the Cuelist play.

For more info on Cuelist, please refer to the Section "**About Cuelists"** in *WYSIWYG Reference Guide*.

Step 9 - 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.

The *Render Wizard* button.

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.

Note: By default, the *Background Rendering Manager* runs on the same system as WYSIWYG. You can also run it on a separate machine. For details on how to set this up, please refer to the *Background Rendering Manager* section of the WYSIWYG Reference Guide.

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

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.

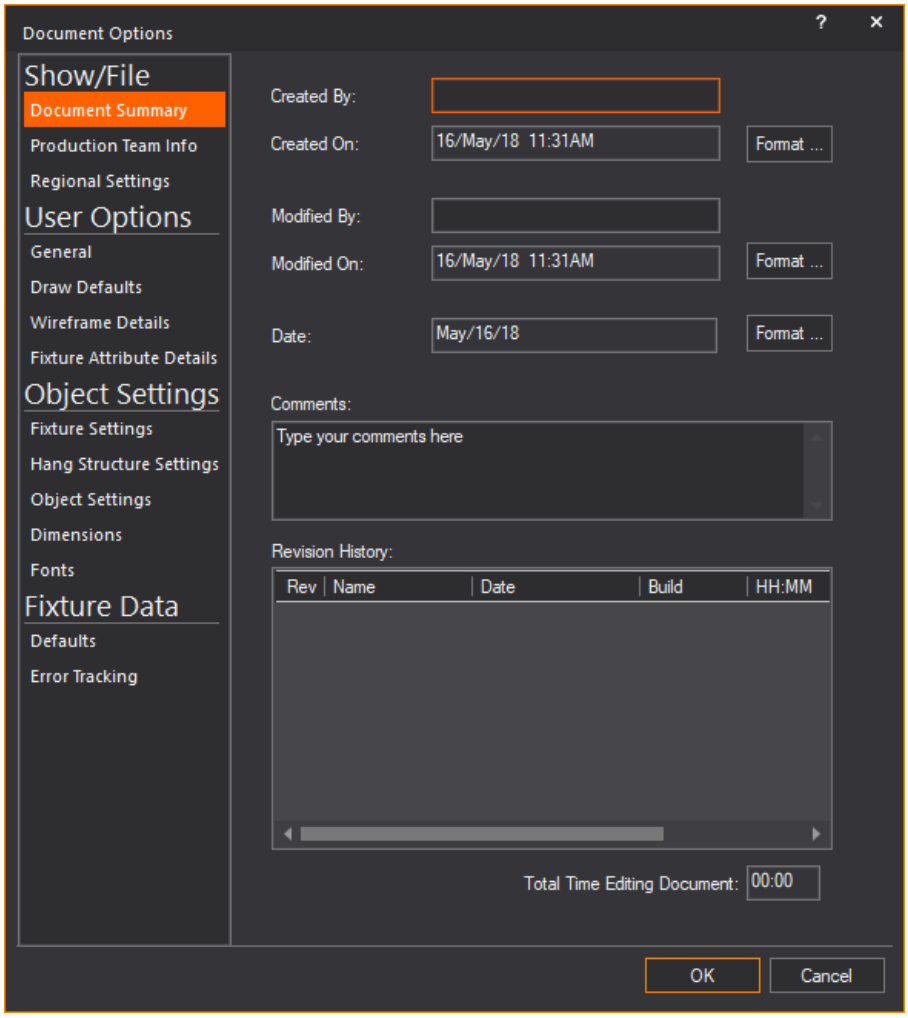
Step 1 - Entering show information

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.

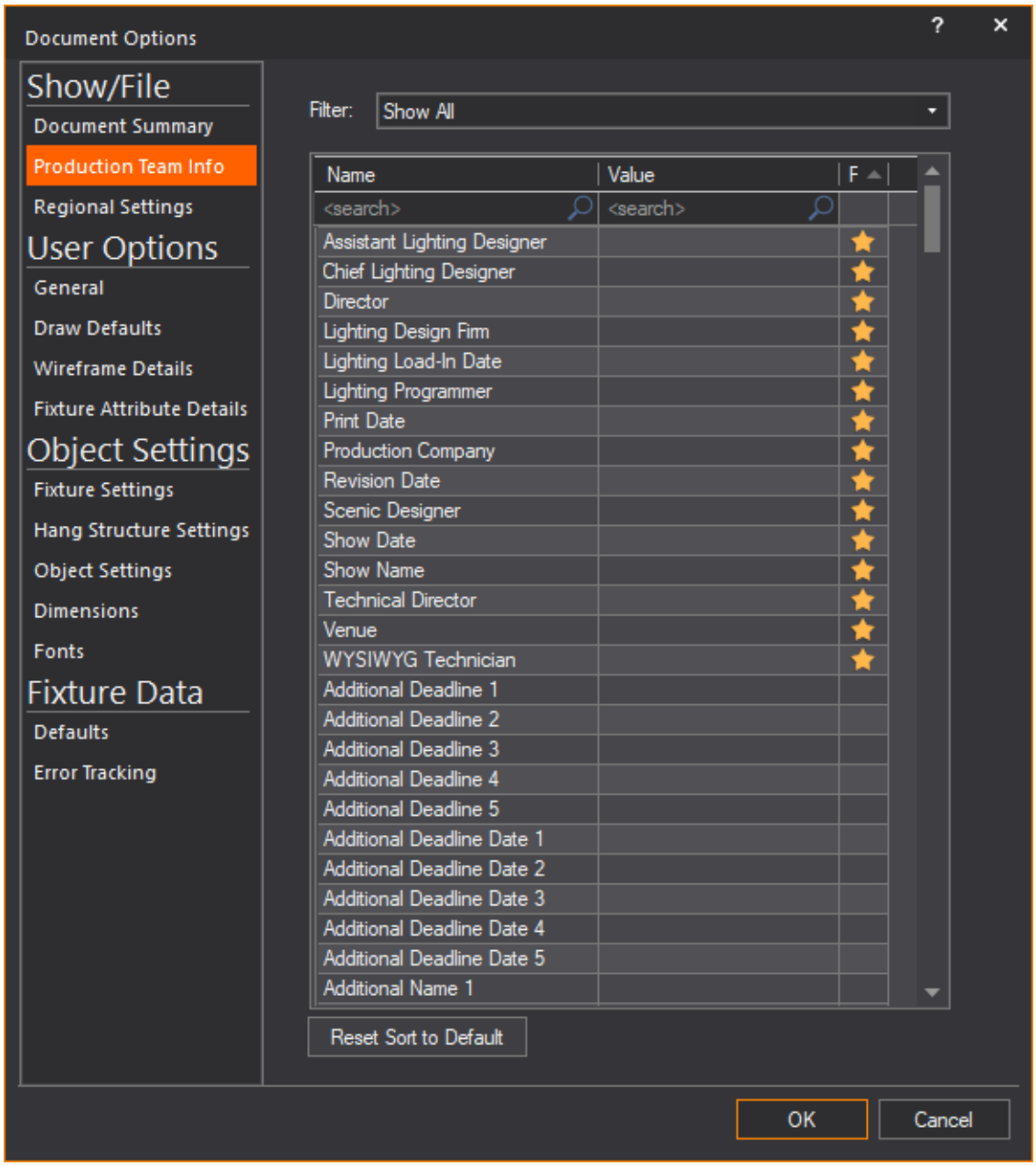
To add or modify show info

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

Result: The *Document Options* window will open.



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*.

Note: Use the Filter above the top to reduce the rows displayed in the table. For example, select "*Show Favorites Only*" to display most commonly used Production Team Info names. Click on the Fav column to add/remove a name from Favorites list.

Step 2 - Modifying a report

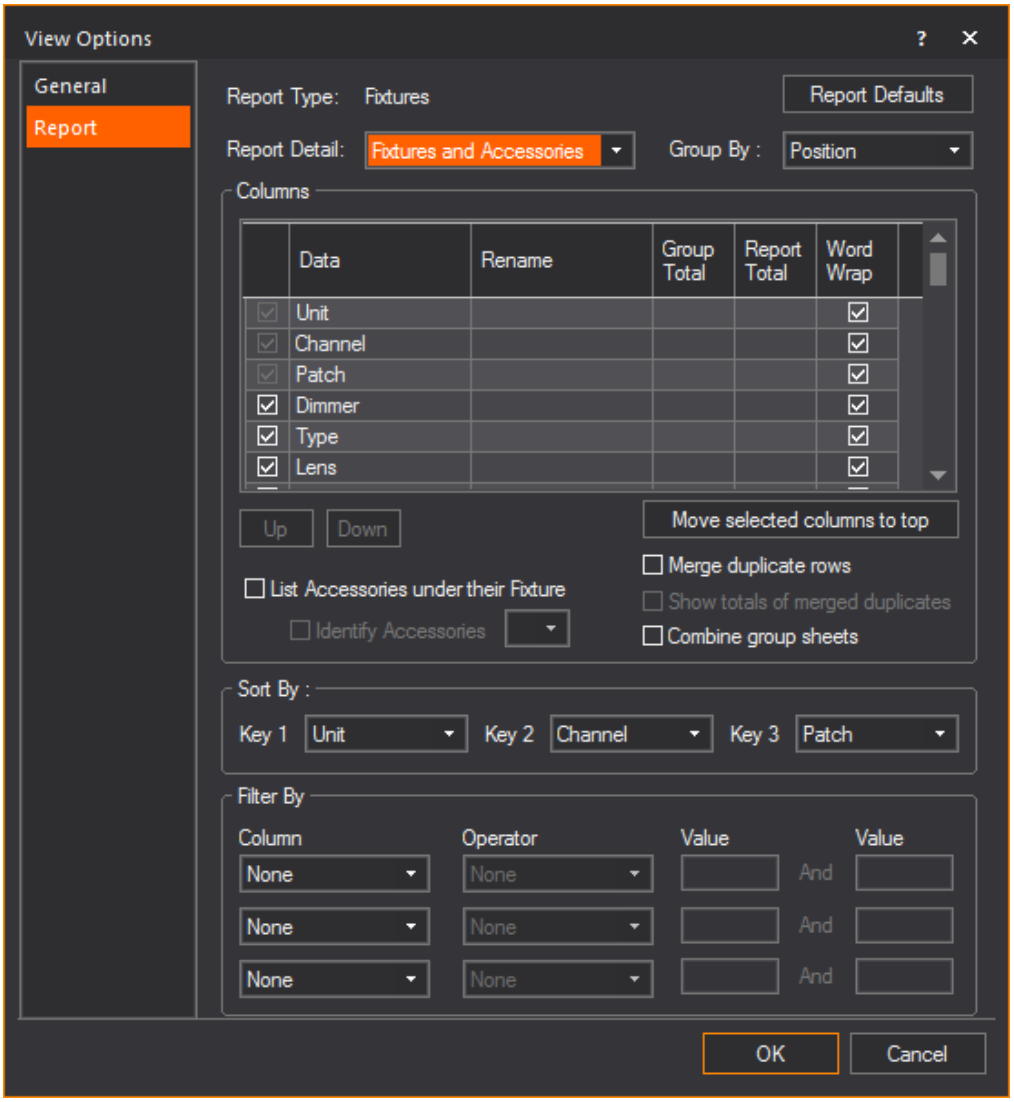
To modify a report

1 In Presentation mode Reports view, in the shortcut bar, 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

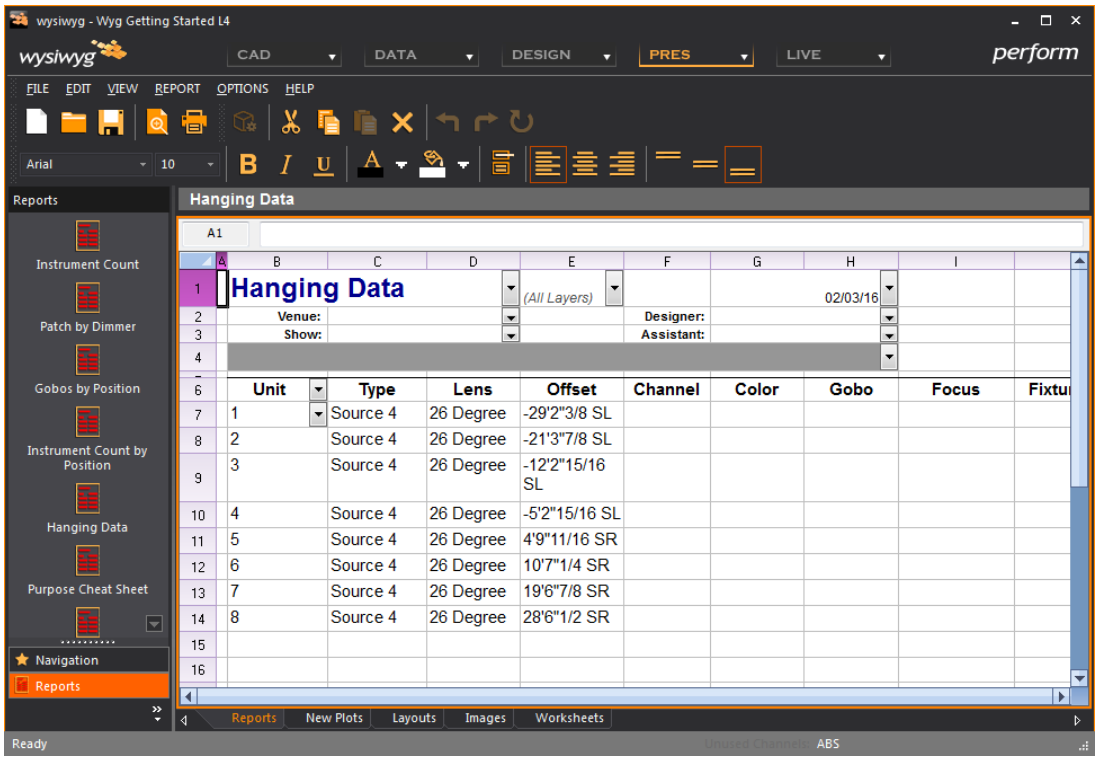
The *Columns* list displays the columns available for inclusion or exclusion in a report. You can display the fields in any order by arranging the order of this list using the *Up* and *Down* buttons. The top of the column list represents the left column on the page and the bottom of the list is the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

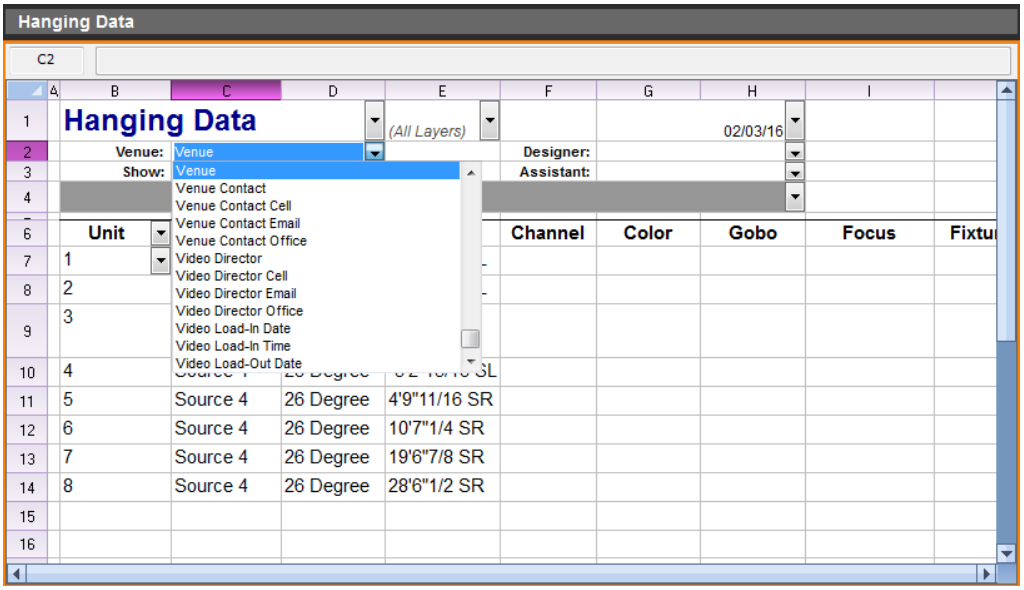
Step 3 - Formatting report headings



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.

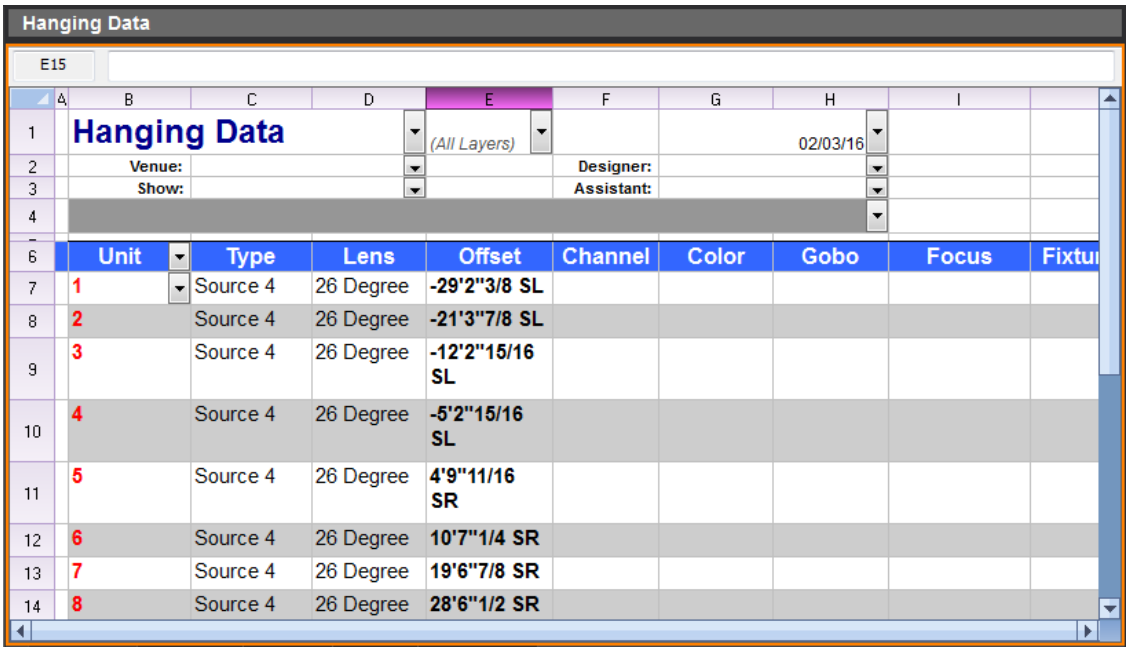
Alternatively, right-click on the data section and select Format Cells.

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.



Hanging Data

E15

	A	B	C	D	E	F	G	H	I
1		Hanging Data			(All Layers)			02/03/16	
2		Venue:				Designer:			
3		Show:				Assistant:			
4									
6		Unit	Type	Lens	Offset	Channel	Color	Gobo	Focus	Fixtu
7		1	Source 4	26 Degree	-29'2"3/8 SL					
8		2	Source 4	26 Degree	-21'3"7/8 SL					
9		3	Source 4	26 Degree	-12'2"15/16 SL					
10		4	Source 4	26 Degree	-5'2"15/16 SL					
11		5	Source 4	26 Degree	4'9"11/16 SR					
12		6	Source 4	26 Degree	10'7"1/4 SR					
13		7	Source 4	26 Degree	19'6"7/8 SR					
14		8	Source 4	26 Degree	28'6"1/2 SR					

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

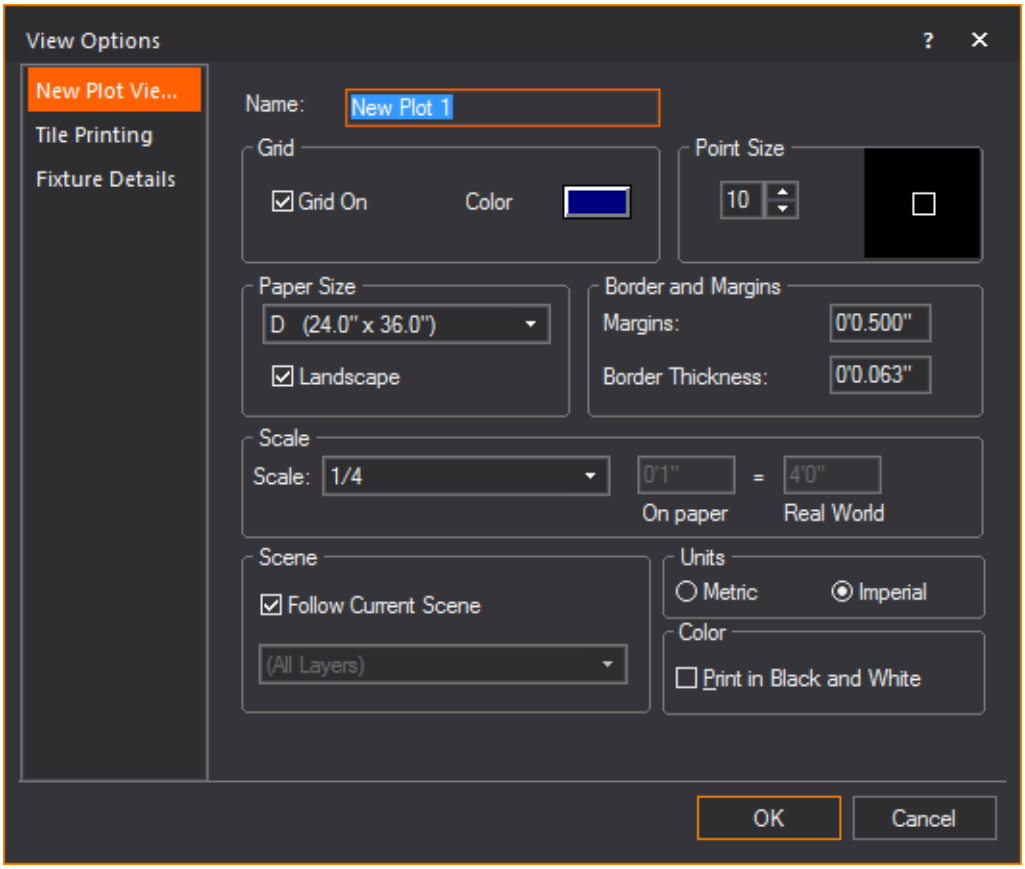
1. In Presentation mode, click the *New Plots* shortcut tab.
2. Right-click in the *New Plots* shortcut area, and then choose the type of plot that you want to create.
3. Type a name, and then click *OK*.

   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**.

Result: The *View Options* window will open.



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

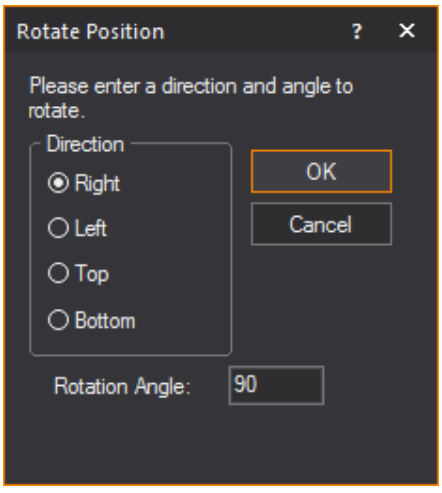
1. In the New Plot, select an object.
2. Click and drag to modify its position; to modify the layer properties, right-click the object, and then choose **Object Properties**.
3. Modify the properties as required.
4. Click *OK*.

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. The *Rotate Position* window will open.



4. Select the direction and angle in which you want to rotate the hang structure.
5. 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.

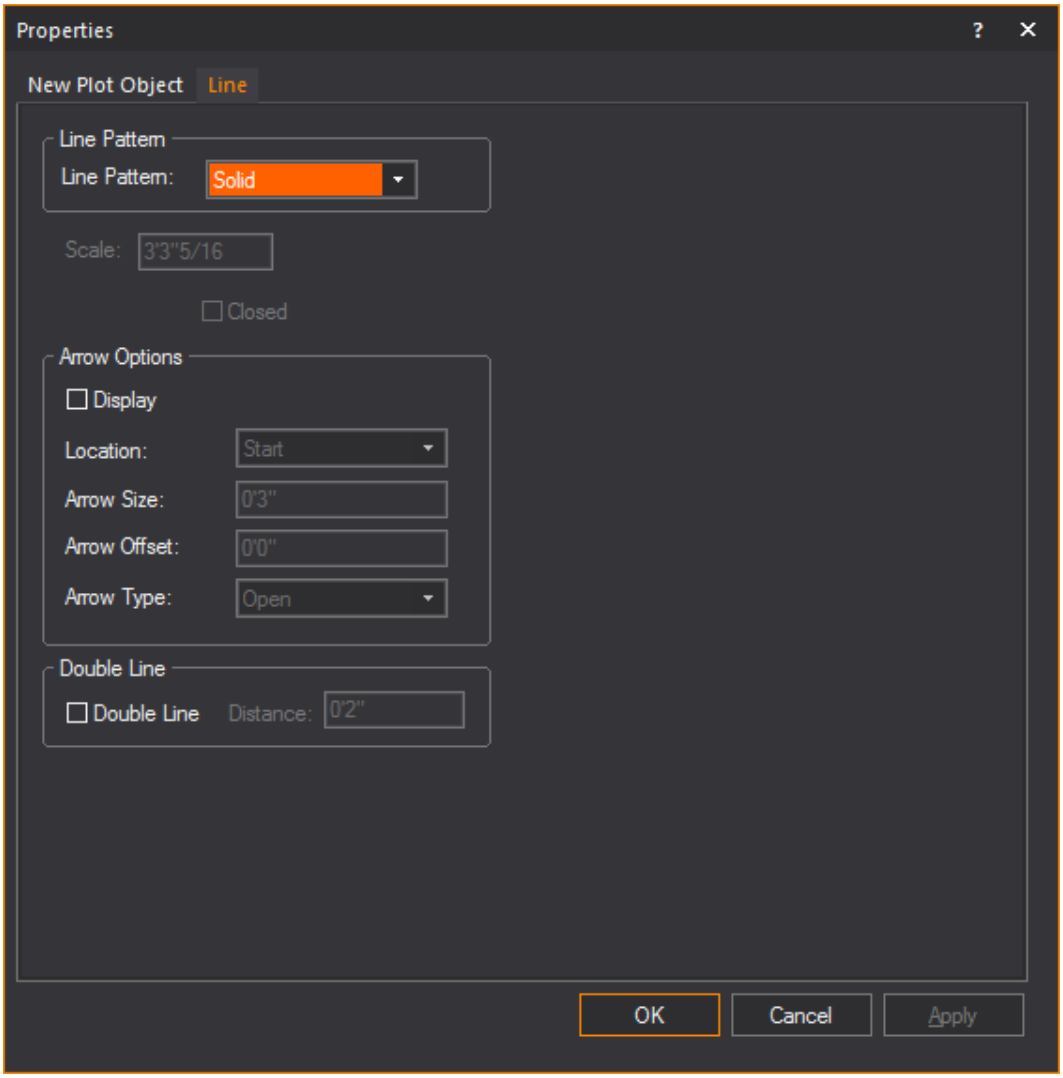
   Tip: You can resize objects by clicking and dragging them.

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**.

   Result: The *Properties* window will open.

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, the *Layouts* tab offers the tools to define all the information you wish to display on your to printed lighting plot. Layouts are created in WYSIWYG by arranging various CAD, report, image, and other items on a defined paper size.

A layout may contain multiple objects/items of the same type. For example, you can place multiple CAD items on the same layout, in order to show the lighting plot from the multiple viewpoints (i.e. Plan view and Side view) or even to show the same viewpoint at different scales (for example, if you need to show the detail of a particularly "dense" or "complex" area of the plot, where the scale of the CAD item simply displays the fixtures, etc., too small to be legible).

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

Step 1 - Creating a new layout

To create a new layout

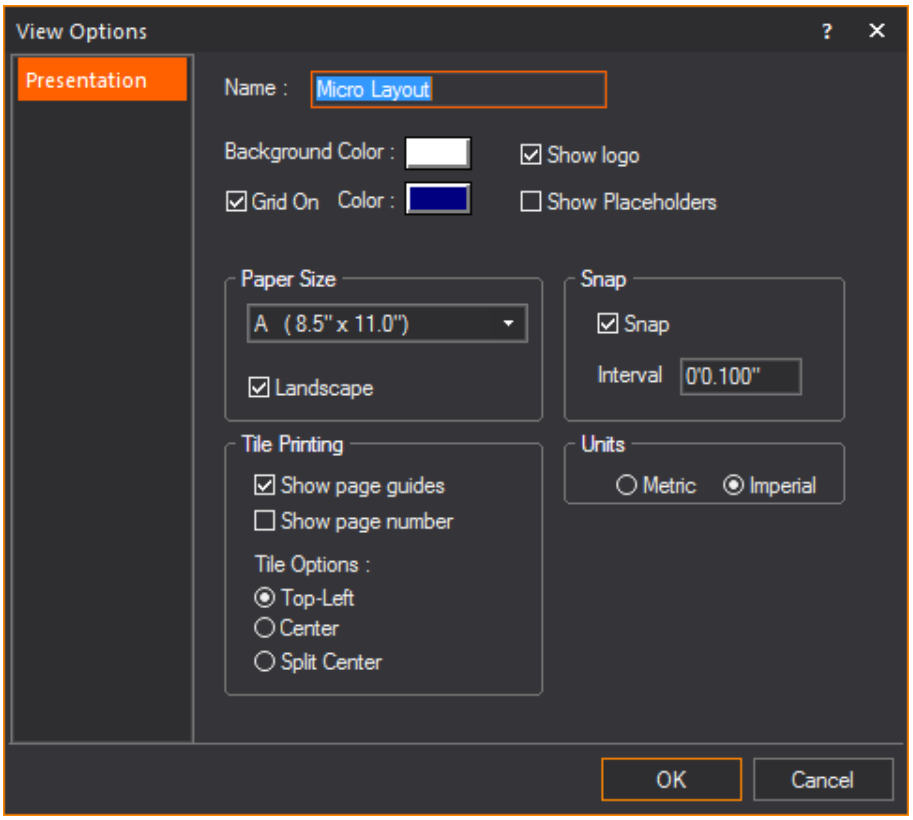
1. In *Presentation* mode, click the *Layouts* tab.
2. Right-click in the *Layouts* shortcut area, and then choose **New Layout**.
3. Type a name, and then click *OK*.

   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 The *View Options* window will open.



3 From the *Paper Size* drop-down list, select the desired paper size.

4 Click *OK* to save your changes and exit.

The Grid and gridlines

When enabled, the Grid option displays gridlines over the paper. These gridlines are constant and for reference only; they assist with the alignment of items on the page. Gridlines do not appear in the printed output.

Snapping

Snapping is available when inserting items on the layout. The snapping interval is defined in the **View Options**, and is completely independent of the gridline spacing.

Paper Size

When choosing a paper size for your layout, select the physical paper size (dimensions) that you will be printing your final plot on. Changing the paper size will change the virtual paper representation, i.e. the white rectangle, displayed in the *Layout* tab which represents your lighting plot.

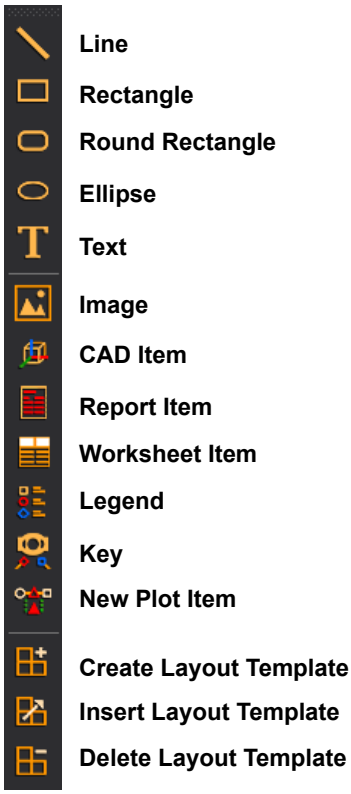
Adjusting Zoom Level and Panning the Layout

Zooming on the layout can be done by pressing the PAGE UP and PAGE DOWN keys on the keyboard, or by scrolling the MOUSE WHEEL. Pan the layout by holding down the MOUSE WHEEL.

Step 2 - Inserting and manipulating 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.

To resize an object in the layout

1 In your layout, click the object you want to resize to select it.

Result: The object's grip handles are displayed.

2 To resize the item on the paper, click and hold on the object's handle, then drag to alter the size.

Locking items in layouts

You can "lock" items in your layouts to avoid accidentally moving or deleting them once you are satisfied with their placement. Once an item is locked, you can still select it (and access its Properties), but you can't move or delete it.

To lock/unlock an object in a layout

1 In your layout, click the object that you want to lock.

2 To lock the object, on the Layout Pres Edit toolbar, click the *Lock* button.

The *Lock* button.

Note: Alternately, you can click CTRL + L to lock the selected item.

Result: The grips of the item turn red instead of black and you cannot move or delete the object.

3 To unlock the object, click on the *Lock* button.

Note: Alternately, you can click CTRL + L to unlock the selected object.

Alignment Tools

Use the Alignment Tools found on the Layout Pres Align toolbar to easily line up objects you've inserted onto a layout.

To align objects

1 On the Layout Pres Align toolbar, set up a point of reference which selected objects will align to. Select one of the following:

Align to Extents - All selected objects (regardless of selection order) will align to the left-most, right-most, bottom-most, etc. point of the left-most, right-most, bottom-most, etc. object, respectively.

Align to First Selected - Objects selected after the first object was selected will align to that first object.

Align to Last Selected - Objects selected before the last object was selected will align to that last object.

2 In the layout, select your first object, and then a second object.

3 Click on an alignment option to align the two selected objects.

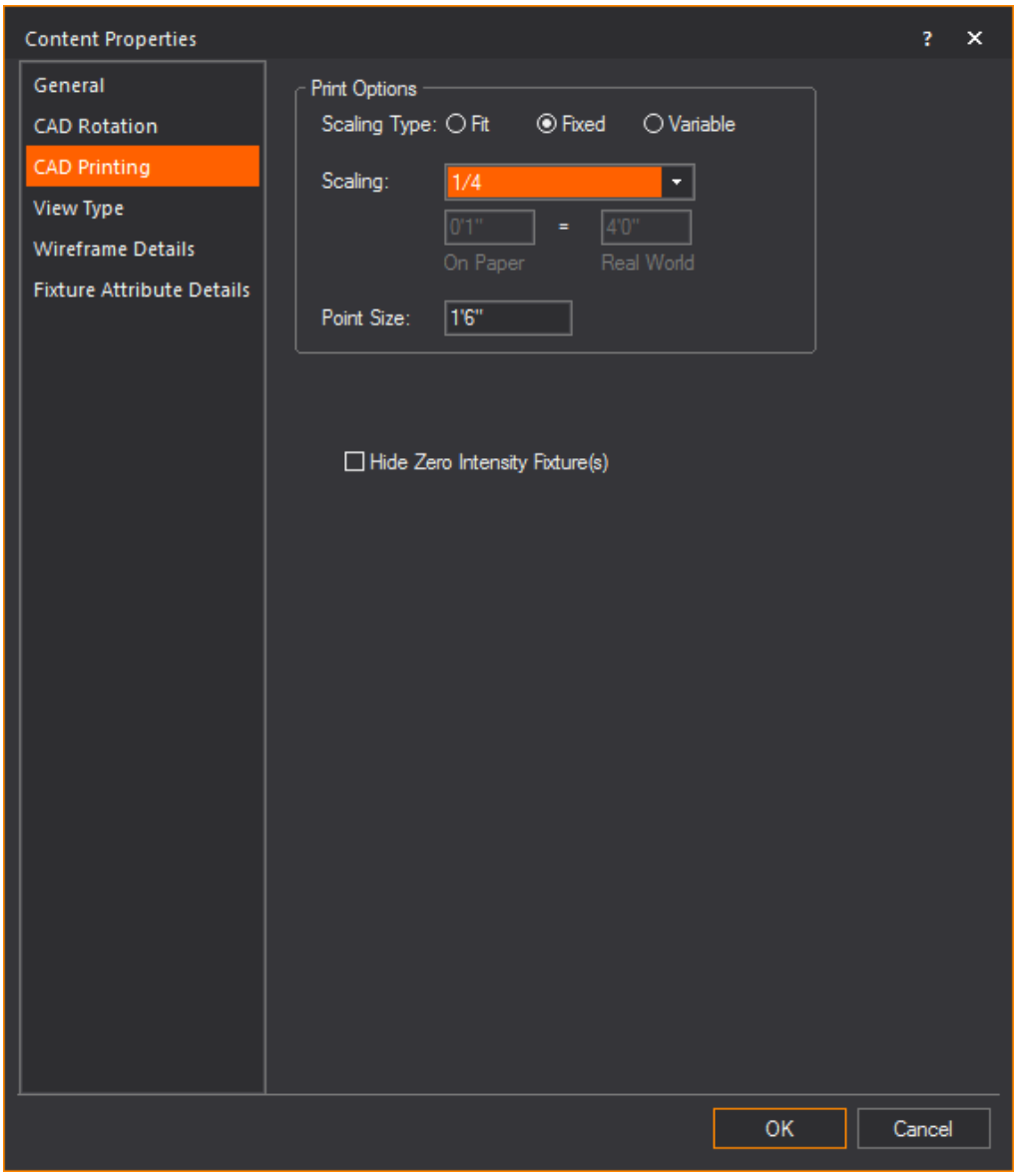
Step 3 - Modifying CAD items

A CAD item contains a section of your wireframe drawing. You can think of it as a 'window' into the CAD mode drawing, on your plot.

To modify CAD items

In this example you will modify the scale of a CAD item. When a new CAD item is added to a layout, its View type is set to Plan view and its scale is fixed to 1/16” = 1’0”. These defaults, and others, may be modified via its *Content Properties* window, as follows:

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



4. Click the *Fixed* option button.
5. From the drop-down list, select *Custom*.
6. Type the scaling to be used when printing the drawing, for example, fixed at 1/8”=1’.
7. Select any other desired items, and then click *OK*.

Step 4 - Adding legends and keys

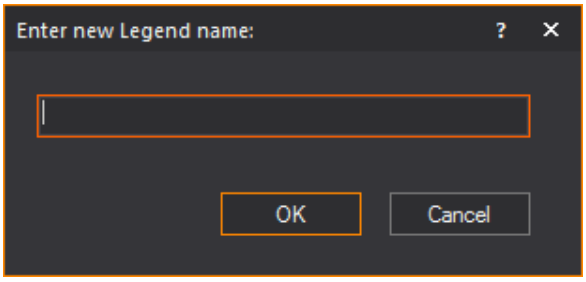
The Legend and Key Wizards guide you through the process of adding a legend or key to your layout.

- **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

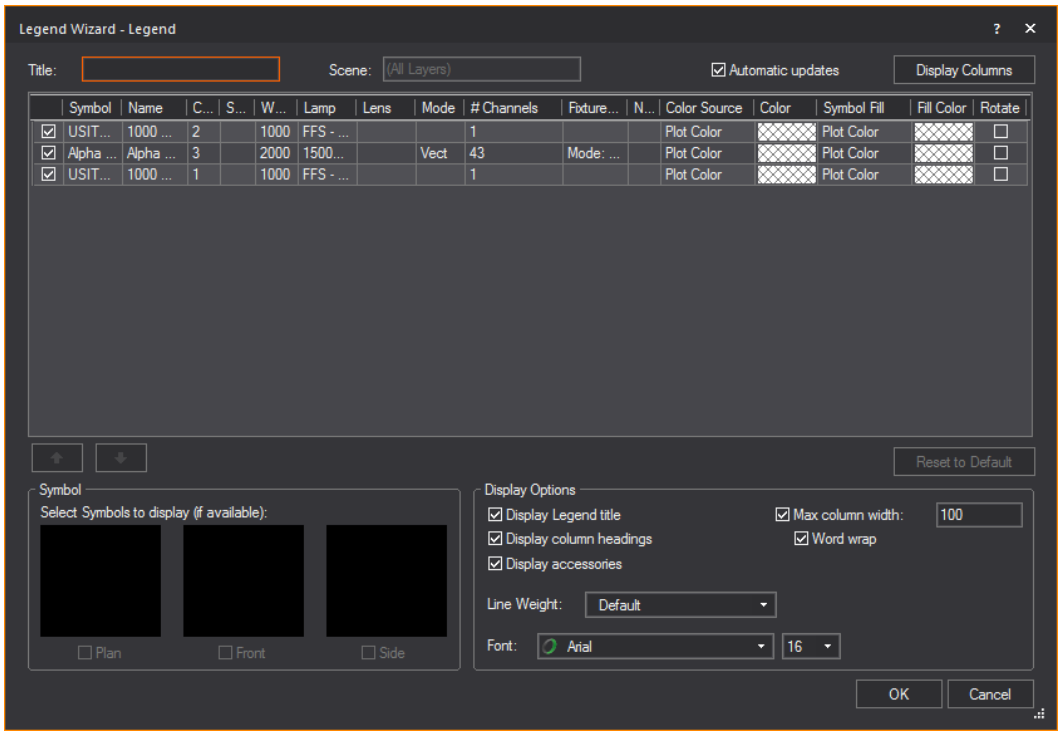
1 Right-click in the *Wizards* shortcut bar, and then select *New Legend*.

Result: The *Enter new Legend name* dialog box is displayed.



2 Type a name for the legend, and then click *OK*.

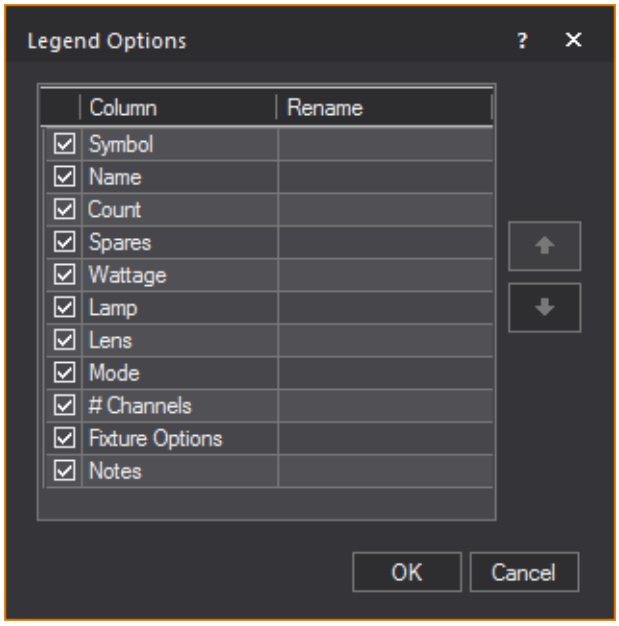
Result: The legend shortcut is stored in the shortcut bar and the *Legend Wizard* opens. The Legend Symbols Editor appears as displayed below. All fixtures in your CAD drawing are listed in the Legend Editor for possible inclusion in the Legend. All the options to configure your Legend are available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



- *Rename*: You can type a name for the selected Legend Column header.
- *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.
- *Mode*: Displays the fixture's mode setting.
- *# Channels*: Displays the total number of DMX channels required by the fixture.

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.

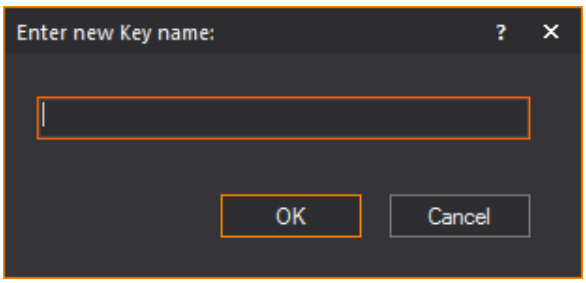
- *Display Legend Title*: Enable this option to display a title for your legend.
- *Display column headings*: Enable this option to display the headings of columns in the legend.
- *Display grid*: Enable this option to show a grid separating symbols in the legend.

- *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*.

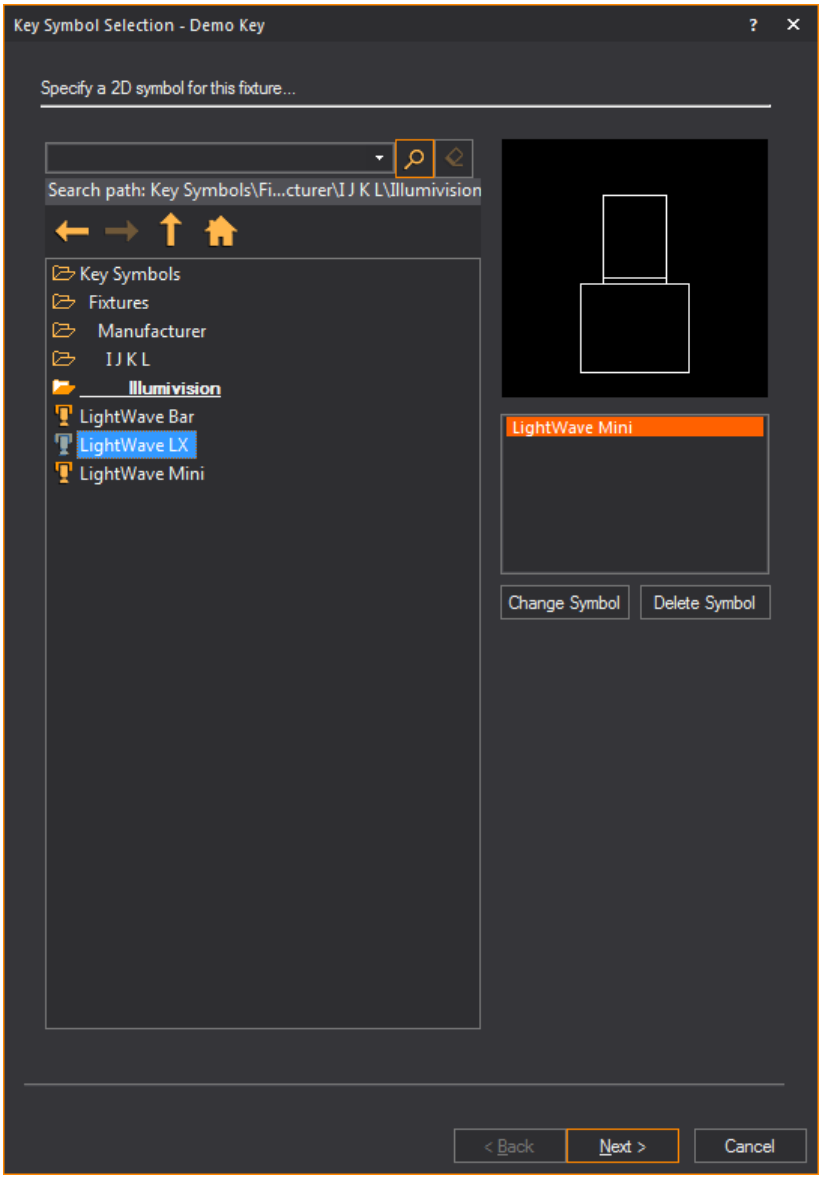
Result: The *Enter new Key name* dialog box is displayed.



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



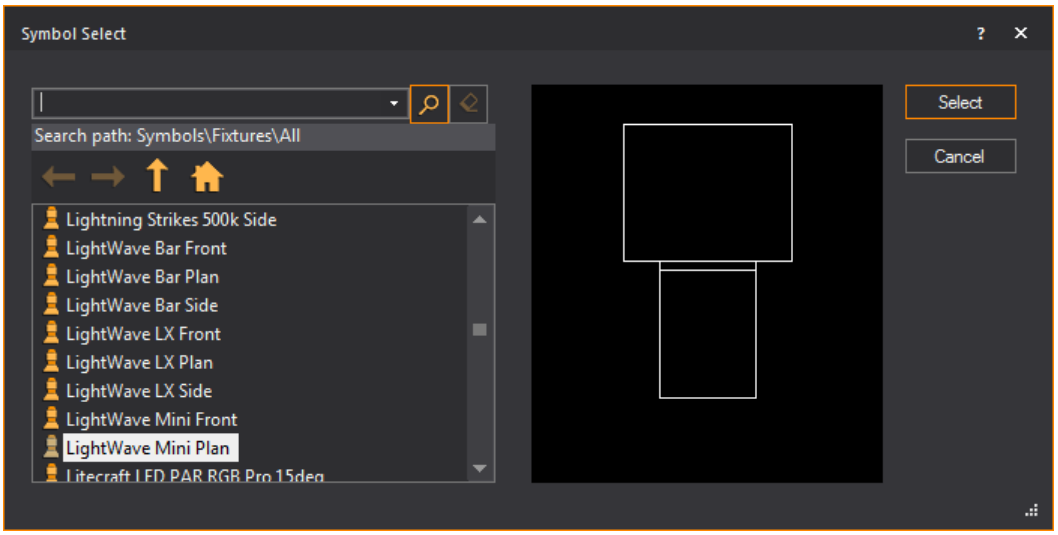
Options in Step 1 allow you to choose the symbol that will represent the fixture or accessory in the key.

- *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.

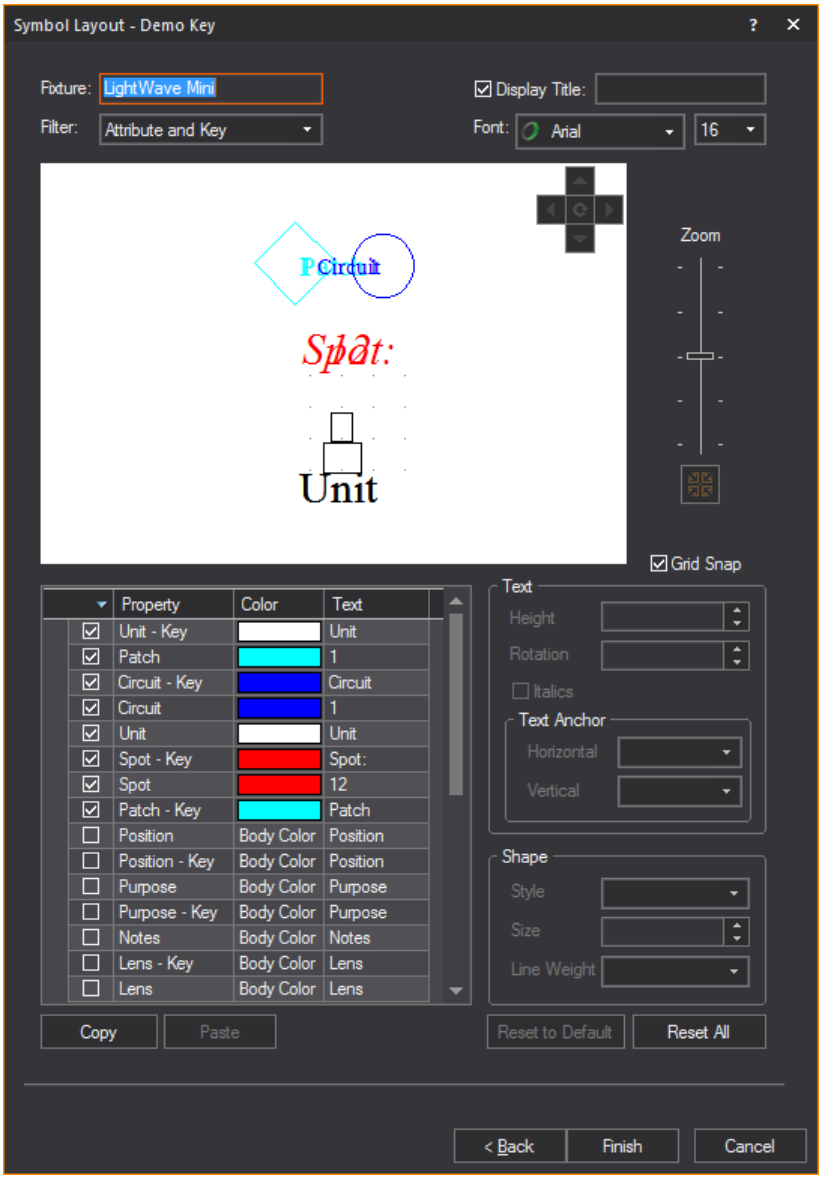
Changing symbols

From the list of items in the window to the right of the preview window, select the symbol. To change the symbol, select the symbol name under the preview window and click the *Change Symbol* button. The *Symbol Select* window 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 tab 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.

Working in the Patch Layout tab

You can view one patch universe at a time. As you add new patch universes in either a Spreadsheet view or a CAD drawing (using Quick Tools), their shortcuts are created automatically and listed in the Patch tab. You can also create new patch universes directly from the Patch tab's Shortcut bar.

Step 1 - Creating a new patch universe in the Patch tab

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", "B", "C", or "Dimmers", "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 universe is created and its shortcut appears in the Patch shortcut bar.

Step 2 - Patching fixtures and movement axes

To patch fixtures and movement axes in the patch layout screen

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

Note: While this method of patching is slow, it is the only method that provides instant visual feedback for what was patched. For faster patching use Quick Tools, or direct input into **DATA** > **Spreadsheets**, as described in earlier lessons.

Step 3 - Repatching/unpatching 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.

To unpatch a fixture

1. Select the desired fixture.
2. In the patch pane, right-click and select **Unpatch**.

   Result: The fixture will be removed from the patch pane and become unpatched.

Step 4 - Reading the patch

Reading the DMX patch

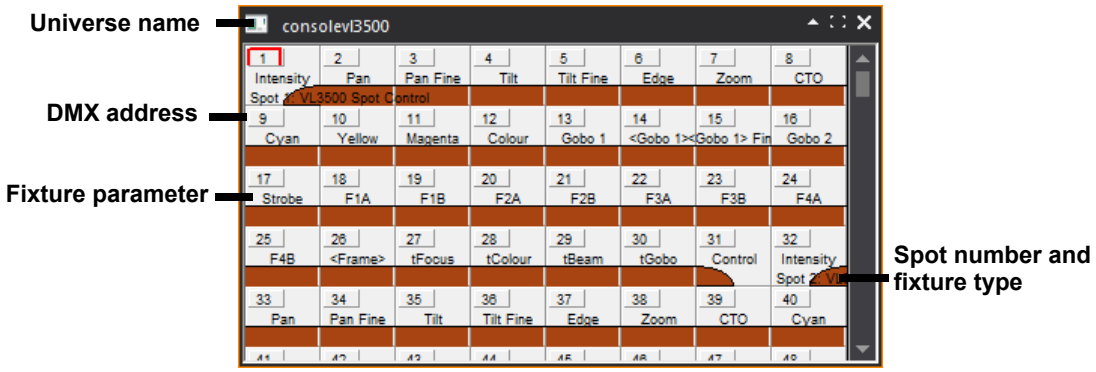
The name of the patch universe is displayed in the title bar.

Each box in the patch universe represents one DMX channel. The channel is identified by the number in the top-left corner. Fixtures are represented by colored bumps that span the number of DMX channels they require. The color of the bump is the Wireframe color assigned to that fixture (typically the fixture's Layer color).

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 fixture parameter controlled by each channel is identified in the center of the box, e.g. Intensity.

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



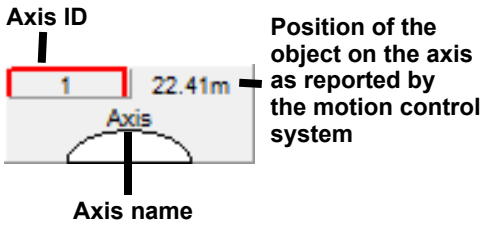
To patch a motion axis to a DMX console device

In WYSIWYG, you can patch a motion axis to which one or more objects has been attached, and then use a console device or a motion control system to control the progress of the objects along the axis.

You have the choice of patching the axis to either a DMX patch universe or a motion control universe, based on the type of console or system you will be using to move the objects. For details on patching the axis to a motion control universe, see "**To patch a motion axis to a motion control system**" in the WYSIWYG Reference Guide.

Reading the patch for moving scenery

If you have patched a motion axis, the patch appears as shown below:



- For linear motion axes, the position of the object is reported in either metric or imperial units (for example, 16m).
- For rotational axes, the value is shown in degrees (for example, 42°).
- If the axis is unrecognized by the motion controller, then the object position is shown as a question mark (**?**).

Note: If the axis type is mismatched between that which you have drawn and the type that you choose in the Motion Control Console (for example, if the actual axis is linear, but you choose rotational in the Motion Control Console), then the cell in patch view turns to red.

For more information on how to patch objects in WYSIWYG, please refer to the WYSIWYG Reference Guide.

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.

WYSIWYG enables you to visualize and pre-cue your show in real time, by connecting to a lighting console, its "on PC" counterpart (running on the same computer as WYSIWYG or another computer) or to a motion control system. All this is accomplished in Live mode.

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

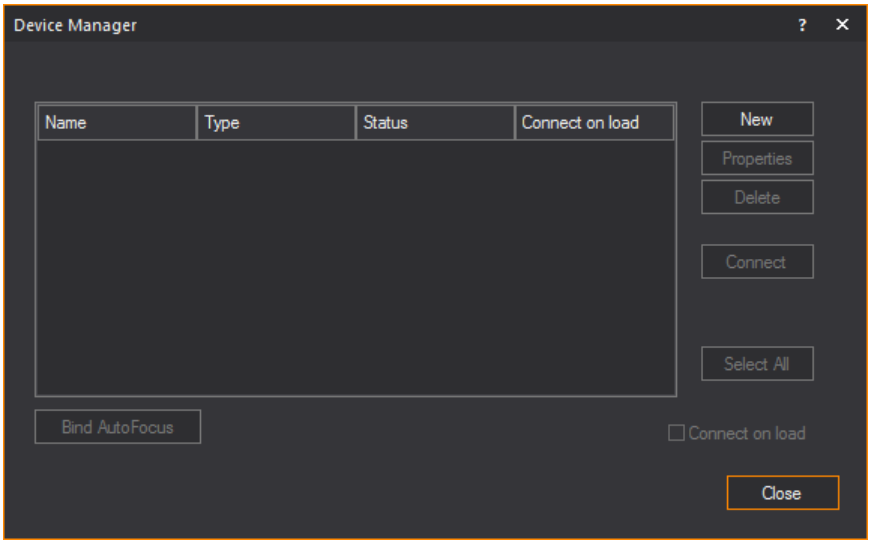
In this lesson you will learn how to insert a console and connect that console in Live mode.

Step 1 - Inserting a console

To insert a console

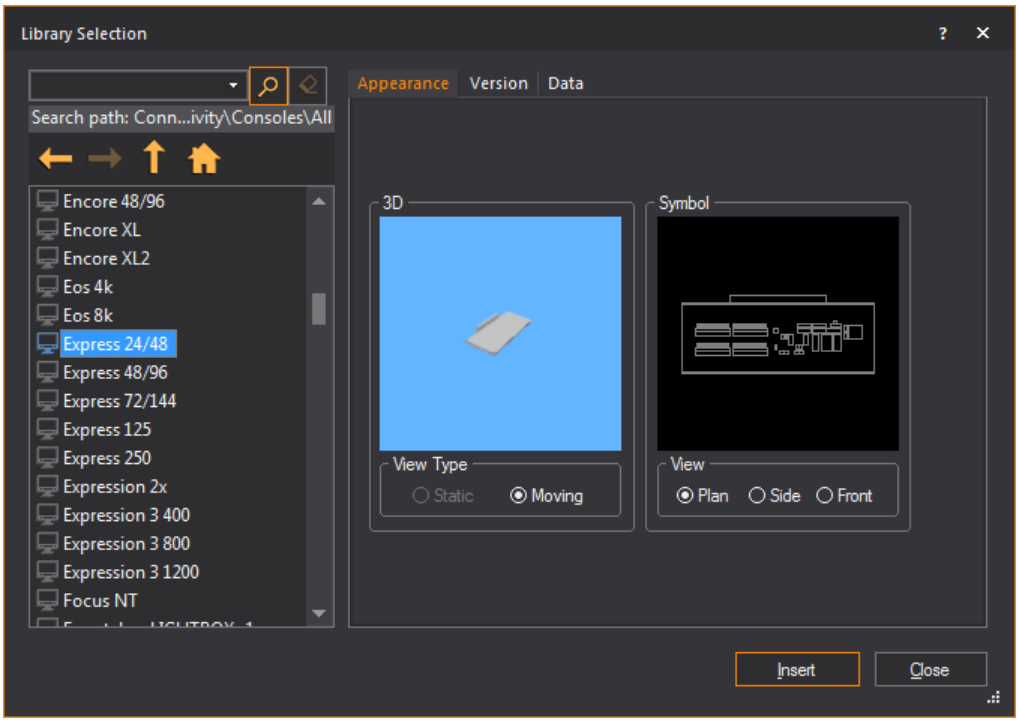
1 From the **Managers** menu, choose **Device Manager**.

Result: The *Device Manager* window will open.



2 In the *Device Manager* window, click *New*.

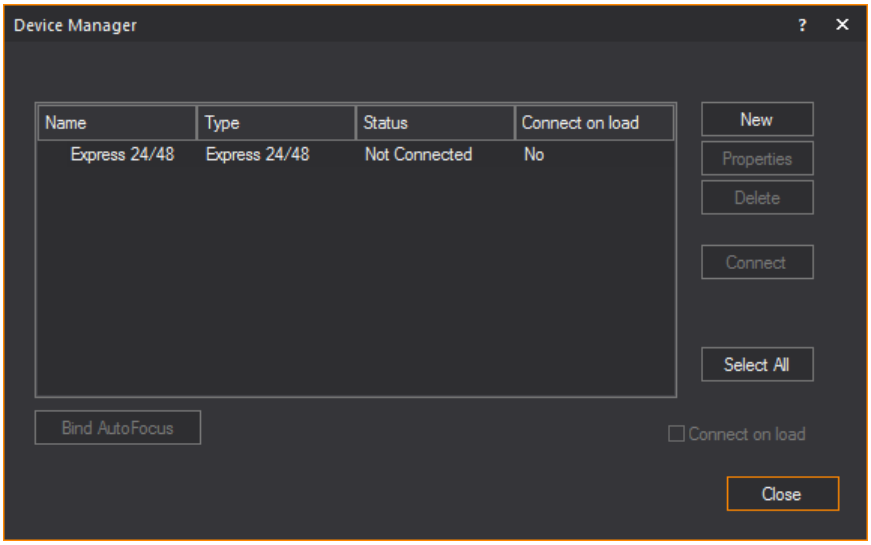
Result: The *Library Selection* window will open.



3 Expand the Manufacturer section, and then choose the desired console (for example, Express 24/48).

4 Click *Insert*.

Result: The console will be inserted into the *Device Manager* window.



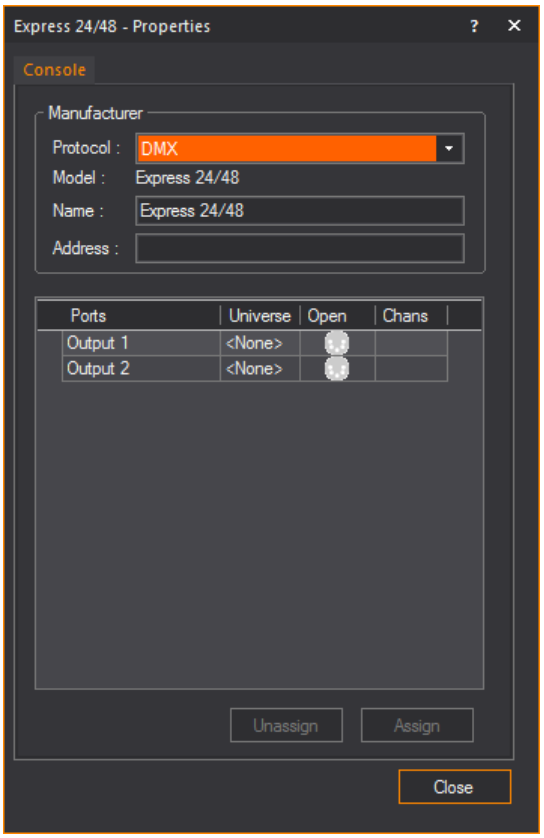
Note: If connecting to a console via a "DMX over Ethernet" protocol such as Art-Net or sACN, the required protocol must be inserted, not the console itself. For more information about connecting to these protocols, see "**Connecting to an sACN Device**" and "**Connecting to an sACN Device"** in the WYSIWYG Reference Guide.

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 The *Properties* window of the console will open.



3 Refer to the connectivity guide for the console you wish to connect to for information on what Protocol and/or Address to select/enter in the *Protocol* and *Address* fields.

4 The *Name* field can be filled with anything at all. For example, "Audience Lighting (Console)", "B-Stage (Console)", etc.

5 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.”

6 Click the *Universe* column to display a drop-down list containing all the patch universes created in this file (Step 1 of Lesson 10). Select from the list the patch universe to bind/connect 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.

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

8 Click *Close* to exit the *Properties* box.

9 Click *Connect*.

Result: The status of the console changes to "connected."

10 Click *Close* to exit the Device Manager.

Note: Once the console is running and connected to WYSIWYG , operate the console as you would normally. 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.

For more information on how to patch objects in WYSIWYG, please refer to the WYSIWYG Reference Guide.

To patch multiple ports simultaneously

Note: When assigning multiple ports to universes, the selected ports will be assigned to universes in sequential order. The universe sequence is based on availability, then alphabetical order.

1 From the **Managers** menu, choose **Device Manager**.

Result: The *Device Manager* dialog box is displayed.

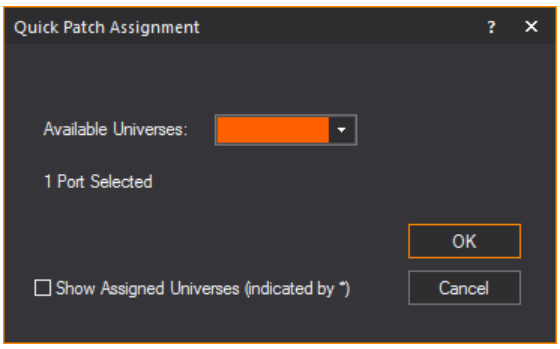
2 Select the console.

3 Click *Properties*.

4 Select the ports for which you want to patch.

5 Click *Assign*.

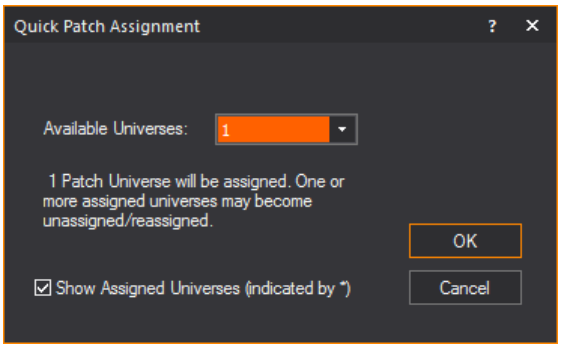
Result: The *Quick Patch Assignment* window will open.



a. To have all universes available for patching, even ones that were previously assigned, mark the checkbox by *Show Assigned universes (Indicated by *)*. Otherwise, only unassigned universe will be available to patch.

6 In the *Quick Patch Assignment* window, from the *Available Universes* drop-down menu, select the universe that will be assigned to the first selected port.

Result: A summary of the assignments will be displayed in the window.



7 Click *OK*.

Result: The ports will be patched to universes. The first port selected will be assigned to the first universe selected. The next sequential port will be patched to the next sequential universe. This continues until all the selected ports are assigned, or there are no more available universes to assign.

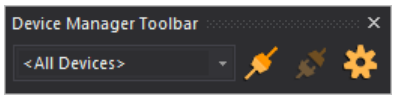
8 Click *Close* to exit the *Properties* box.

9 Click *Connect*.

Result: The status of the console changes to "connected."

10 Click *Close* to exit the Device Manager.

Device Manager toolbar



The *Device Manager* toolbar simplifies your regular device connect/disconnect operations. You can use the drop-down on this toolbar to select a device (Consoles, Devices, Networks), and then click the connect icon or the disconnect icon.

The "All Devices" option in the drop-down enables you to easily connect/disconnect all devices with one click without opening the *Device Manager.*

For more information on connecting to a console or device, or to connect to a "DMX over Ethernet" network, please refer to the appropriate sections in the WYSIWYG Reference Guide.

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.

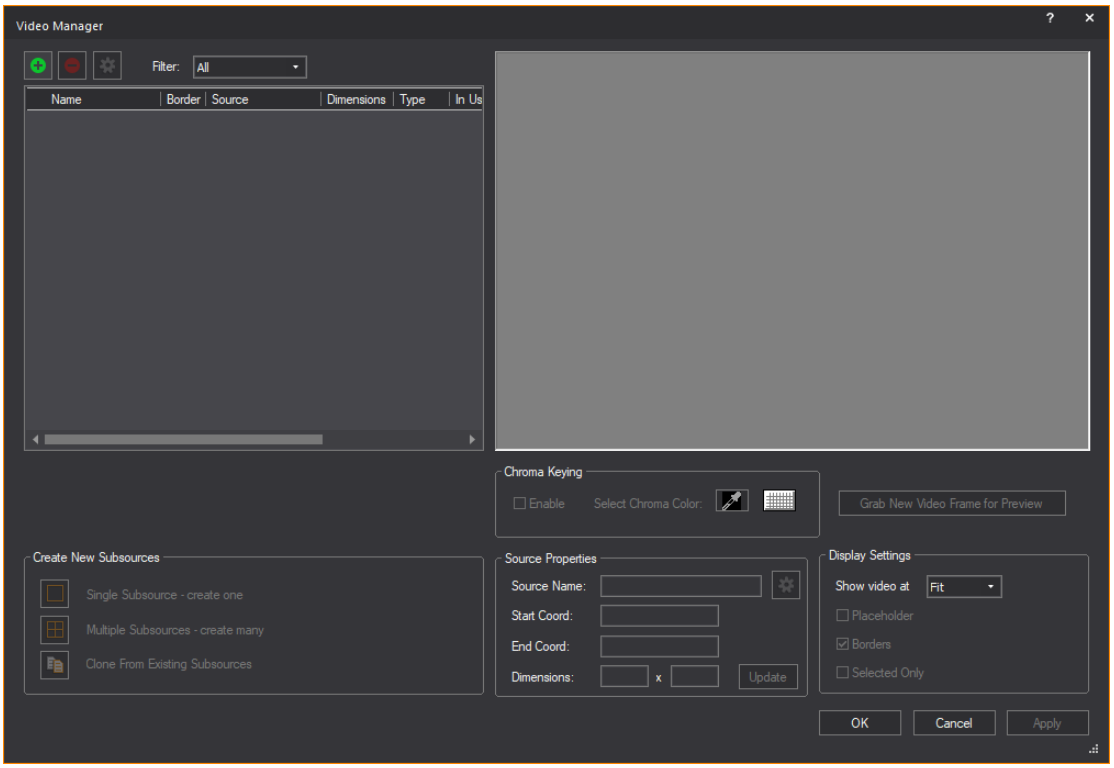
Notes: If you cannot view these file types, you may not have the proper CODEC installed on your PC. Install the appropriate CODEC, and then try viewing the file again.

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. In LIVE mode, click **Managers** > **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.

4 In the **Input** section, select and configure the new video source.

- 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.

- 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.
- Use the Network Device Standard to stream video in real-time from an external device within the IP space. NDI® is a standard that allows multiple video systems to encode, transmit and receive streams of high quality, low latency, frame-accurate video over an

IP in real-time. For more information on NDI®, click *More Info* to see the link http://https://www.ndi.tv/ and access basic NDI® tools.

Note: NDI® video streams will be detected when they are in the same logical network as your WYSIWYG computer.

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 In CAD mode, 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*.

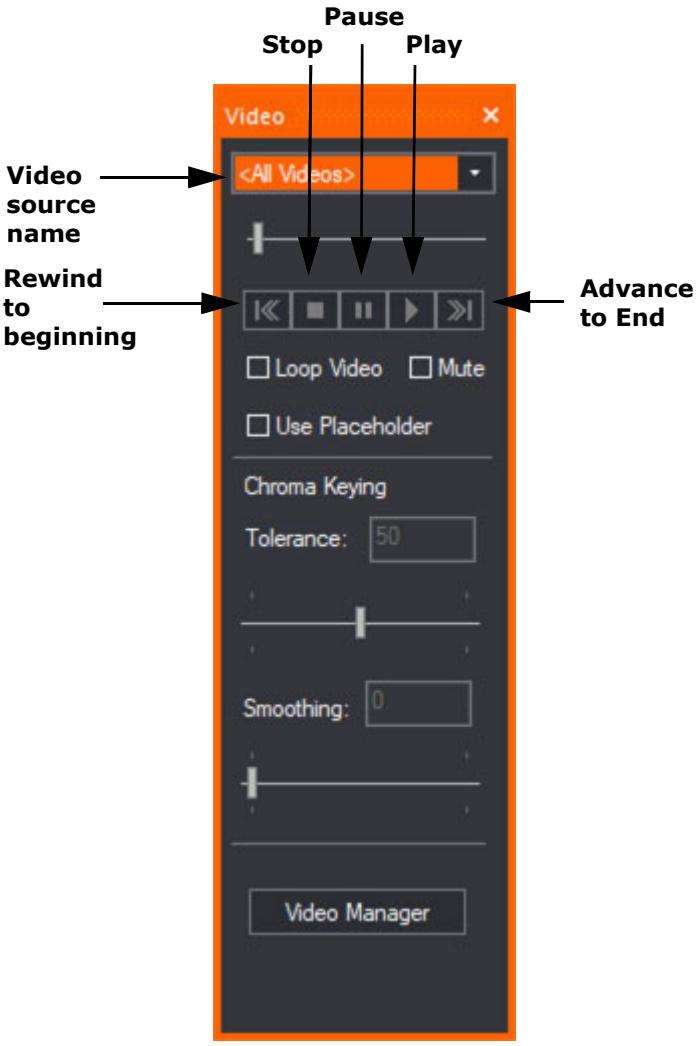
Step 3 - Using the Video design tool to play the video

After you draw the screen and attach the video source, you use the *Video Designer Tool* in Design or CAD 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 *Video Design 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.

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 in the Patch tab" on page 95.

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 in the Patch tab" on page 95.

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/Frame* 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* button.

The *Moving Scenery Designer Tool* button.

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.

The *Fade Looks* button.

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 118.

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 101.
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 101.
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*.

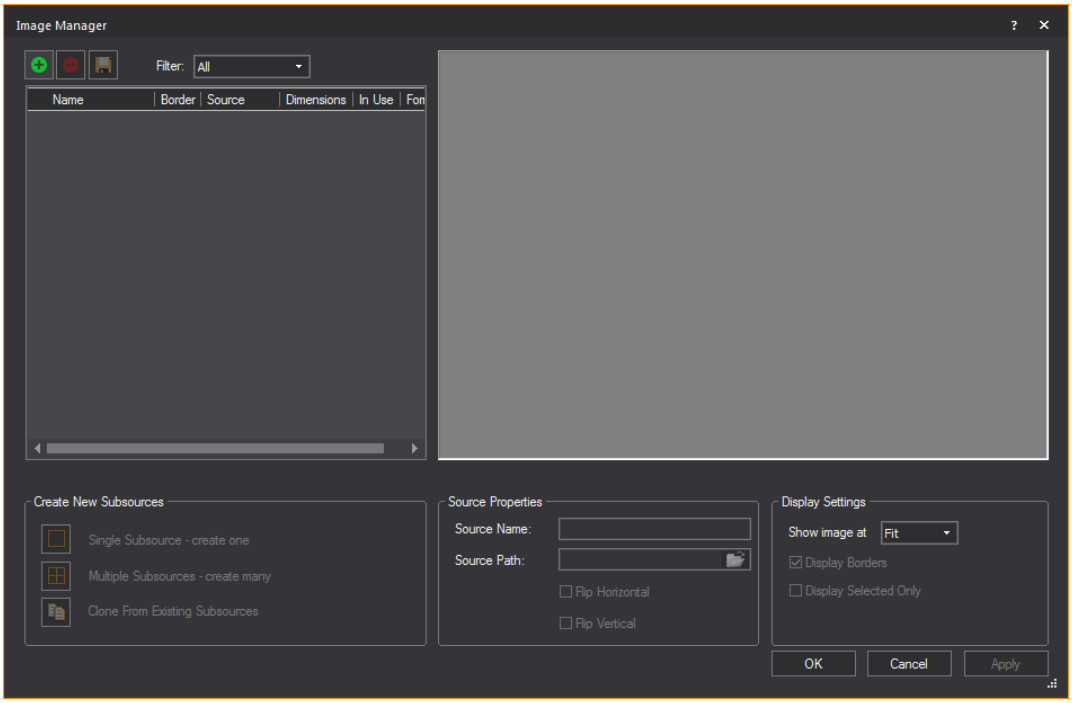
You can use the *Image Manager* to create subsources of an image (i.e., divide the image up), which can then be applied to objects such as surfaces, screens, LED walls.

Step 1 - Creating an image source in the Image Manager

To access the Image Manager

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

Result: The *Image Manager* window appears.

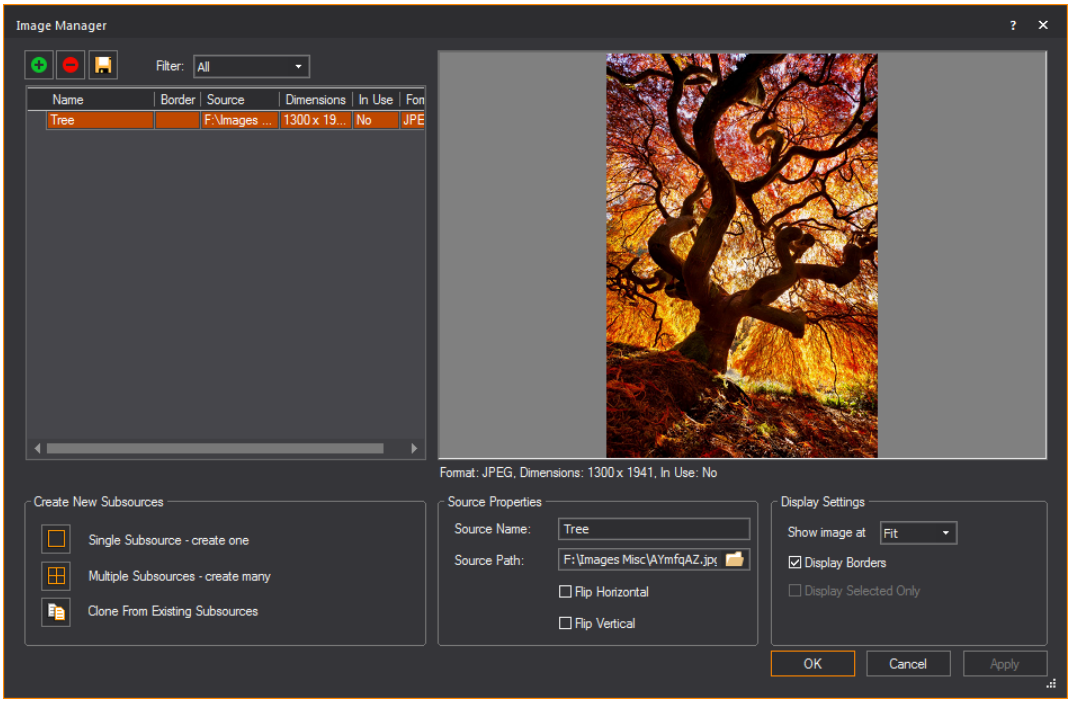


To create a new image source in the image manager

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* button to select a new file.

   The *File Open* button.

- *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*).

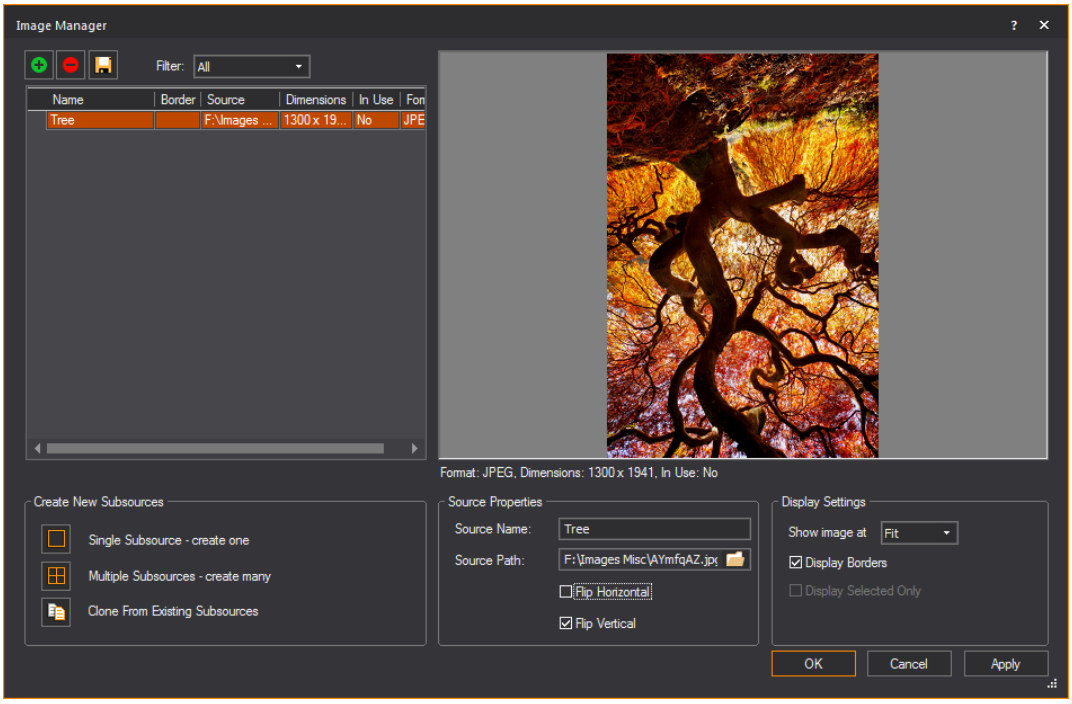
Step 2 - Flipping and exporting image sources

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.

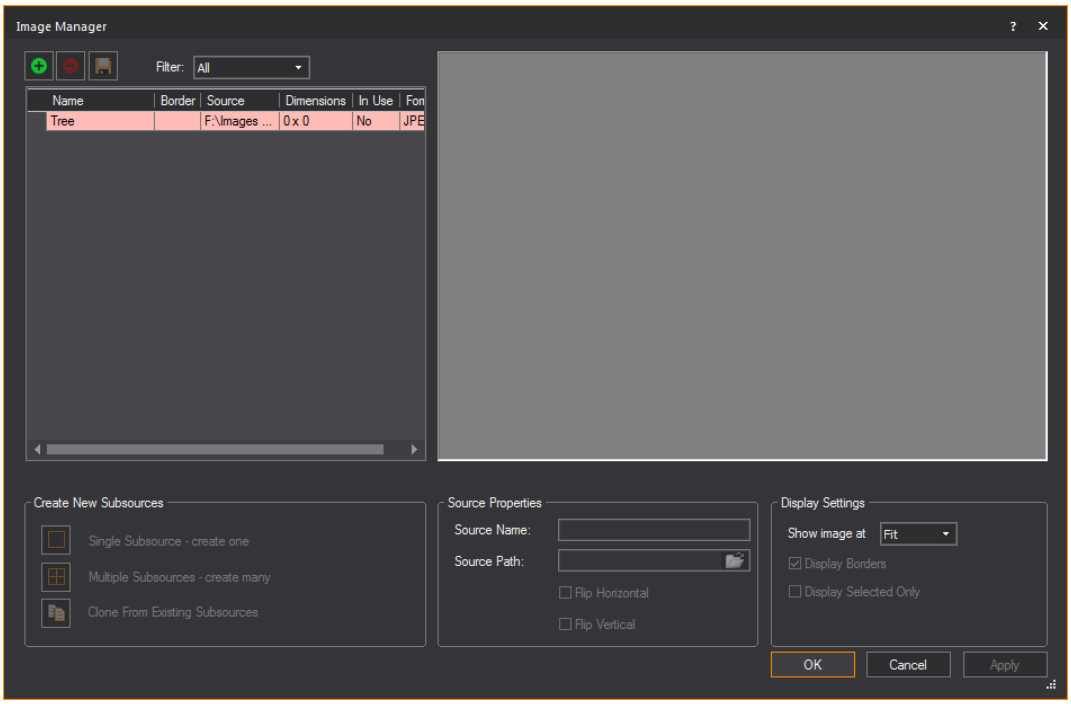
4 Type a name and select a file type for the new image, or accept the default file type of PNG.

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*.

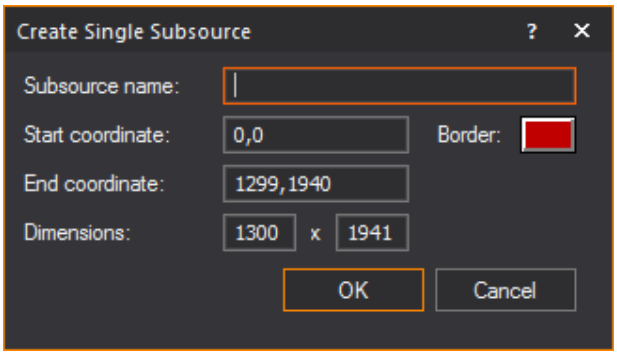
Step 4 - Creating image subsources

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*.

   Result: The *Create Single Subsource* window will open.



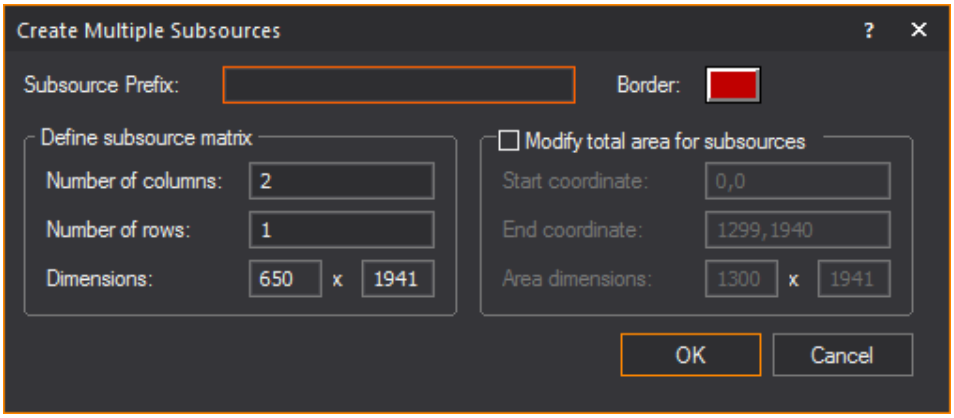
3. Type the subsource name.
4. By default, the system uses the start and end coordinates of the selected source, but you can change these values as desired. When you change these values, the system automatically updates the dimensions.
5. Click the *Border* box to specify the color of the border that 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*.

Result: The *Create Multiple Subsources* window will open



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 properties of 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*.