

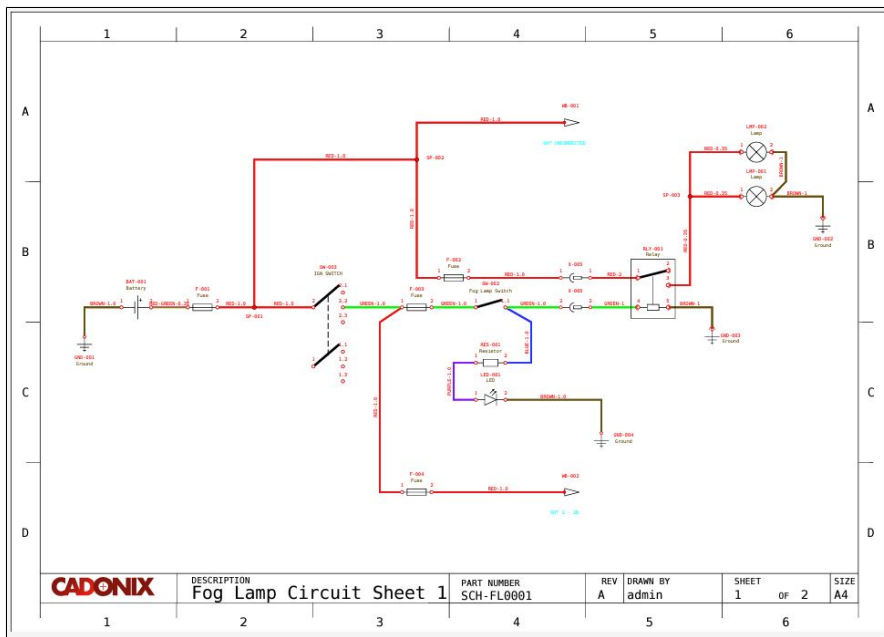
## Introduction

Welcome to Arcadia, the most technologically advanced CAD solution for Harness design available. Built to be powerful yet easy to use, Arcadia enables users to become productive rapidly.

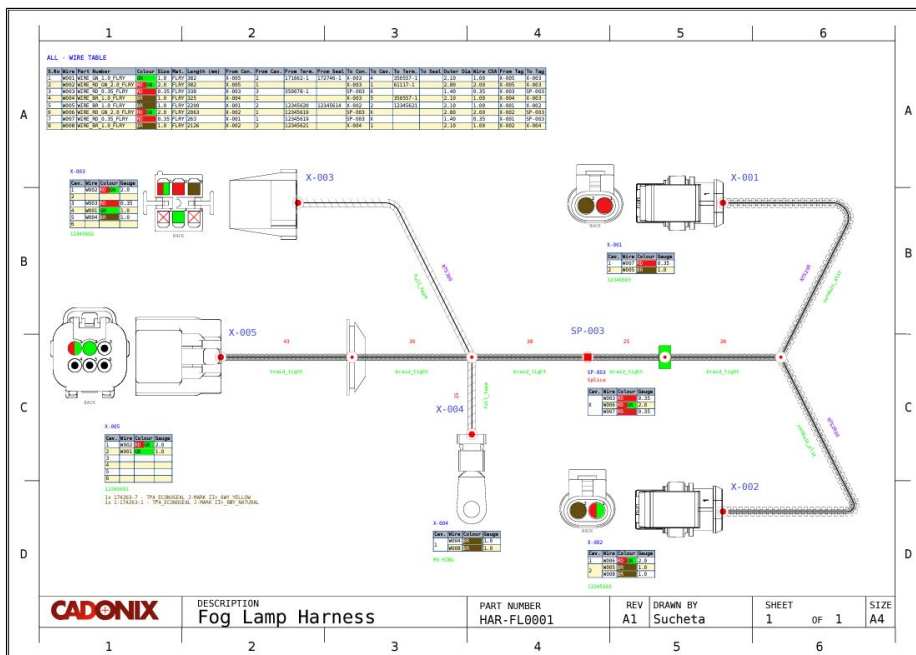
By following this short guide you will successfully complete the Fog Lamp Schematic and Harness Layout shown below.

When you complete this Quick Start Guide, you will have experienced the capabilities of Arcadia, including:

- The ease in which electrical schematics can be created
- The power of performing simulation on the electrical circuit
- The speed and validation of automatic component selection
- The built in design rule checking and automatic creation of harness reports



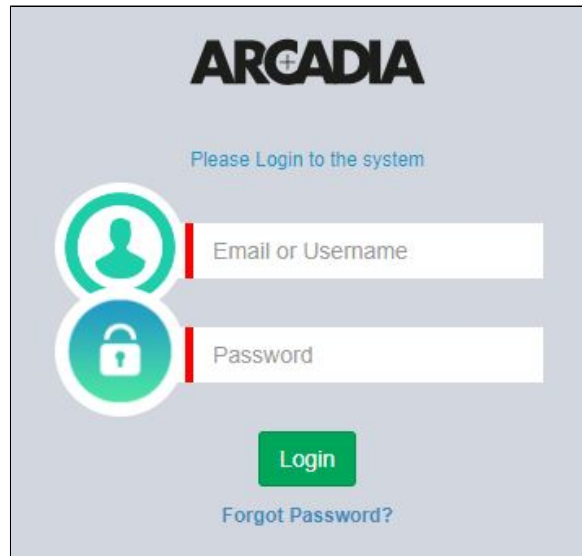
*Fog Lamp Schematic*



*Fog Lamp Harness Layout*

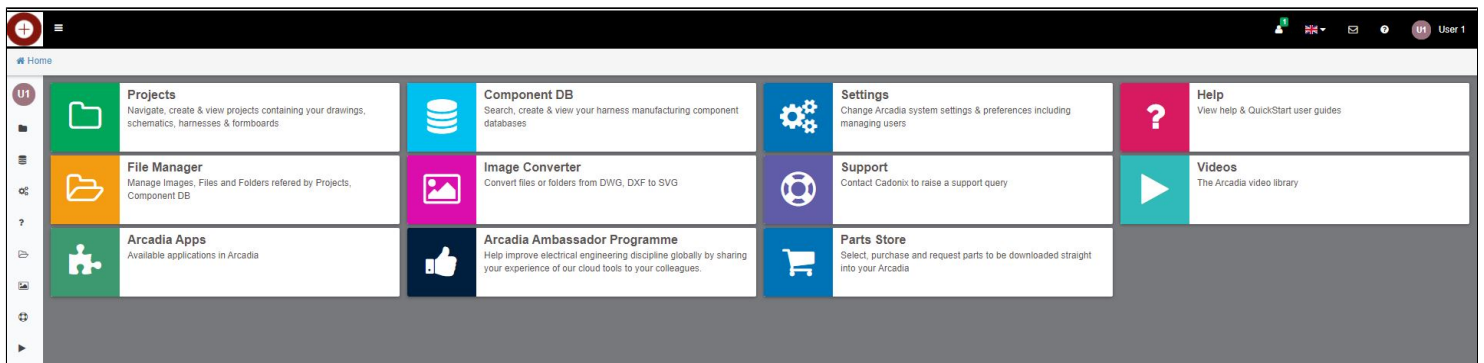
## 1. Getting Started

Firstly navigate to the Arcadia login page and enter your Username and Password (**Fig 1.1**). Click **Submit**.



*Fig 1.1 Arcadia Login Screen*

Once logged in, you are presented with the Arcadia Home screen (**Fig 1.2**).



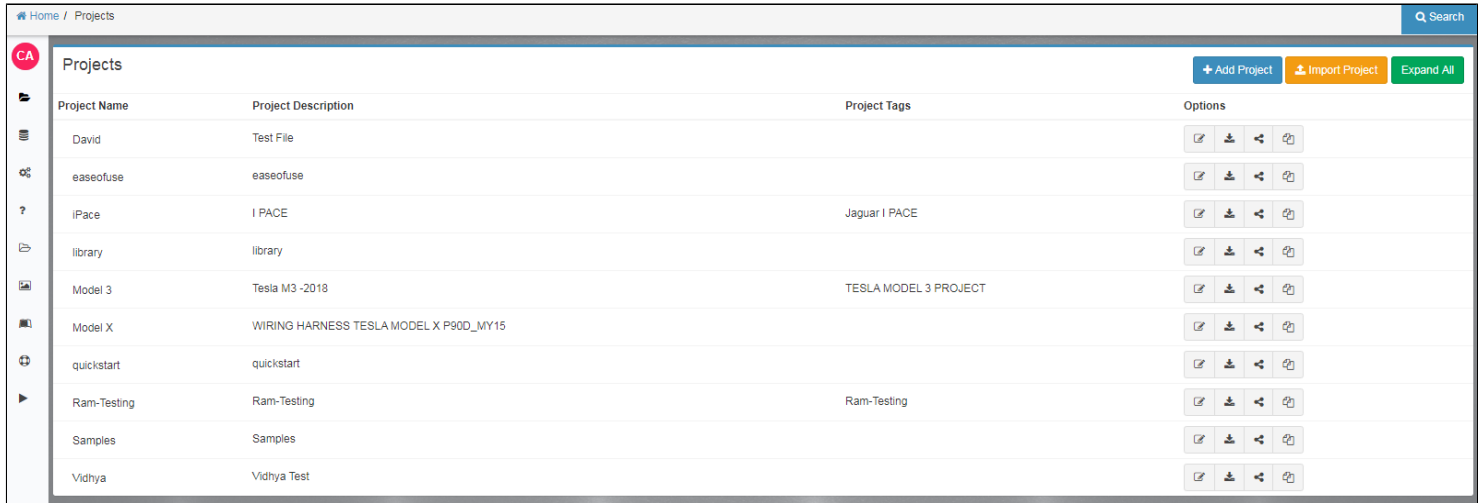
*Fig 1.2 Arcadia home screen*

From here you have access to your Projects, Component Library, Settings, Help, File Manager, Image Converter, Technical Publications, Videos, As well as your Arcadia Apps and the Arcadia Parts Store. You can also change the language settings.

## 2. Open a Project

To begin, you will open an existing Project. In Arcadia, each Project can store your Schematic Harness and Drawing related data.

Click the **Projects** icon on the Arcadia Home screen. You are now in the **Project Manager**.

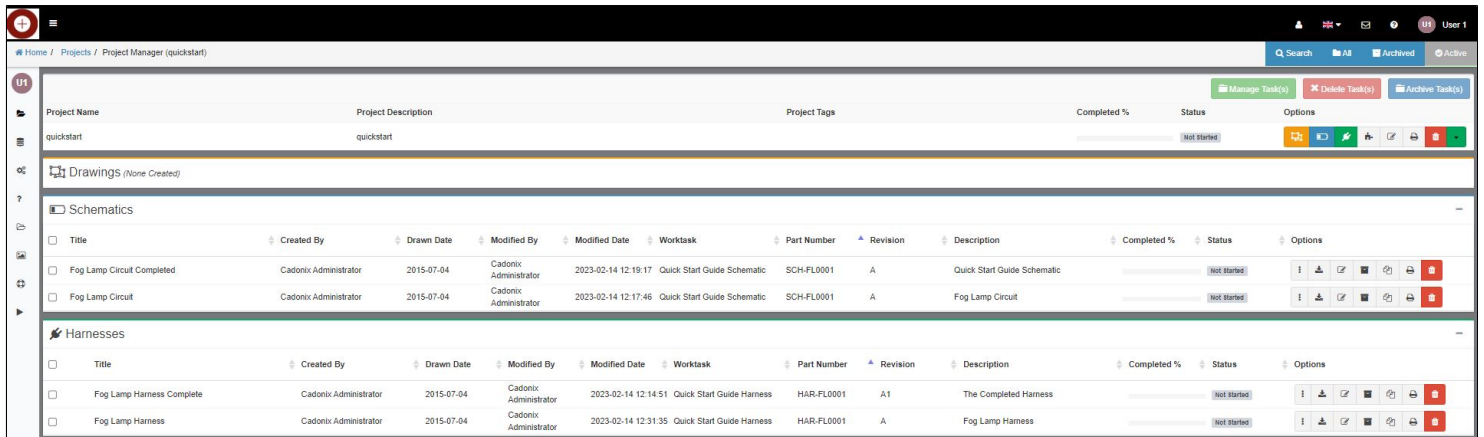


The screenshot shows the 'Projects' page in the Arcadia Project Manager. The page has a search bar at the top right and three buttons: '+ Add Project', '+ Import Project', and 'Expand All'. Below these is a table with the following columns: Project Name, Project Description, Project Tags, and Options. The table contains the following data:

Project Name	Project Description	Project Tags	Options
David	Test File		[Icons]
easeofuse	easeofuse		[Icons]
IPace	I PACE	Jaguar I PACE	[Icons]
library	library		[Icons]
Model 3	Tesla M3 -2018	TESLA MODEL 3 PROJECT	[Icons]
Model X	WIRING HARNESS TESLA MODEL X P90D_MY15		[Icons]
quickstart	quickstart		[Icons]
Ram-Testing	Ram-Testing	Ram-Testing	[Icons]
Samples	Samples		[Icons]
Vidhya	Vidhya Test		[Icons]

Fig 2.1 The Project Information.

Locate the **Quick Start** and click to enter. You are now looking at the **Project Details** screen for the Quick Start Guide.



The screenshot shows the 'Project Manager (quickstart)' page. The top navigation bar includes 'Home / Projects / Project Manager (quickstart)', a search bar, and buttons for 'Manage Task(s)', 'Delete Task(s)', and 'Archive Task(s)'. The main content area is divided into three sections: 'Project Information', 'Drawings (None Created)', and 'Schematics'. Below 'Schematics' is a table with columns: Title, Created By, Drawn Date, Modified By, Modified Date, Worktask, Part Number, Revision, Description, Completed %, Status, and Options. The table contains the following data:

Title	Created By	Drawn Date	Modified By	Modified Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Circuit Completed	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:19:17	Quick Start Guide Schematic	SCH-FL0001	A	Quick Start Guide Schematic		Not started	[Icons]
Fog Lamp Circuit	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:17:46	Quick Start Guide Schematic	SCH-FL0001	A	Fog Lamp Circuit		Not started	[Icons]

Below the 'Schematics' table is a section for 'Harnesses' with a similar table structure. The table contains the following data:

Title	Created By	Drawn Date	Modified By	Modified Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Harness Complete	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:14:51	Quick Start Guide Harness	HAR-FL0001	A1	The Completed Harness		Not started	[Icons]
Fog Lamp Harness	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:31:35	Quick Start Guide Harness	HAR-FL0001	A	Fog Lamp Harness		Not started	[Icons]

Fig 2.2 The Drawing Information Within the Project.

### 3. Creating & Importing Frames

The library project is a standard project in Arcadia.

The library project is a special project because it stores the default drawing frames used in Arcadia. You should not delete this project.

#### Create/Modify a Drawing Frame

You can create or modify a drawing frame in Arcadia.

Open the "**library**" project and click on the drawing title "**frames**". Drawing frames are listed on the left. Each sheet represents a new drawing frame. Clicking on one of the sheets will open the corresponding sheet.

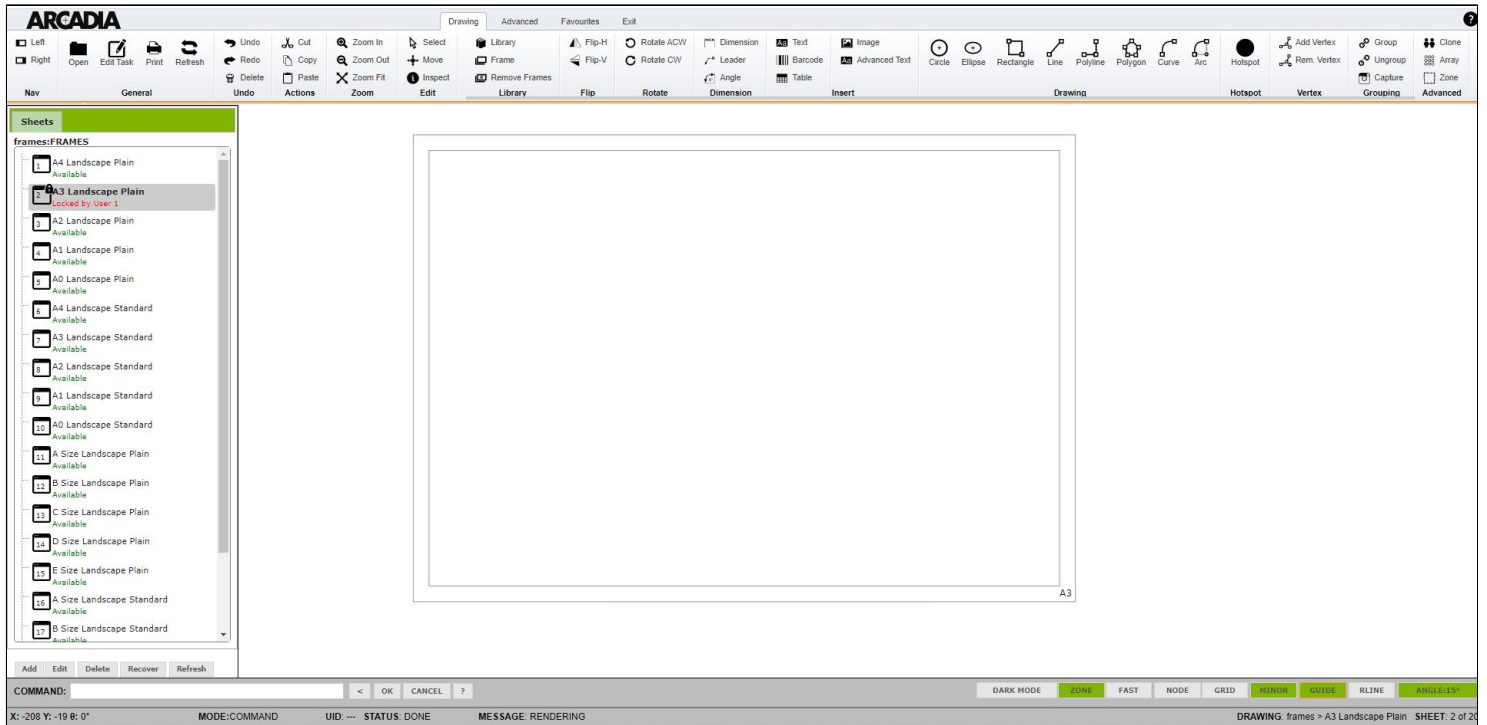


Fig 3.1 Frames

Use the Drawing & General Commands to modify an existing drawing frame.

Create a new drawing frame by clicking on "**Add Sheet**" and entering a description for the drawing frame. On the newly create sheet use the Drawing & General Commands on to draw a new drawing frame.

You can delete a drawing frame at any time by clicking on "**Delete Sheet**".

Now you will draw a basic layout of a frame.

---

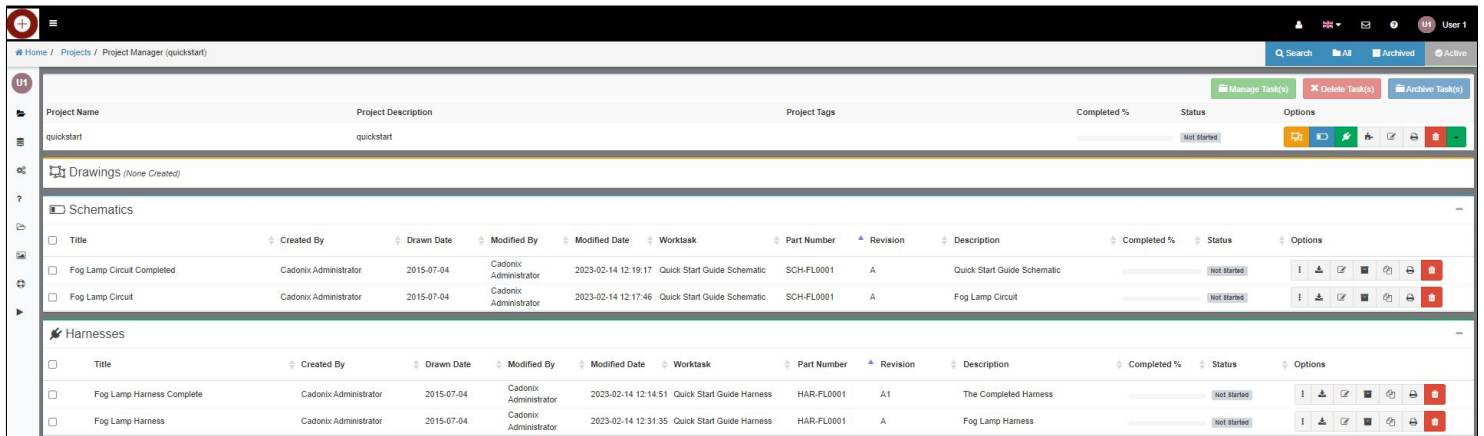
## Schematic Design Quick Start

Using the Arcadia toolset, you will learn to finish the Fog Lamp Schematic and then simulate the design to create a complete electrical prototype.

→ **Note:** The subsequent topics are included as part of the following Arcadia editions: **Arcadia Schematic Professional**, **Arcadia OEM Bundle** and **Arcadia Harness Bundle**.

## 4. Open a Schematic

Click on the title **Fog Lamp Circuit** to open the Fog Lamp Schematic.



The screenshot shows a software interface for project management. At the top, there is a navigation bar with 'Home / Projects / Project Manager (quickstart)' and a search bar. Below this, there are three main sections: 'Drawings', 'Schematics', and 'Harnesses'. Each section contains a table of items with columns for Title, Created By, Drawn Date, Modified By, Modified Date, Worktask, Part Number, Revision, Description, Completed %, Status, and Options. The 'Schematics' section is currently selected and shows two entries: 'Fog Lamp Circuit Completed' and 'Fog Lamp Circuit'. The 'Fog Lamp Circuit' entry is highlighted, indicating it is the item to be opened.

Project Name	Project Description	Project Tags	Completed %	Status	Options
quickstart	quickstart			Not Started	

Drawings (None Created)											
Title	Created By	Drawn Date	Modified By	Modified Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Circuit Completed	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:19:17	Quick Start Guide Schematic	SCH-FL0001	A	Quick Start Guide Schematic		Not Started	
Fog Lamp Circuit	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:17:46	Quick Start Guide Schematic	SCH-FL0001	A	Fog Lamp Circuit		Not Started	

Harnesses											
Title	Created By	Drawn Date	Modified By	Modified Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Harness Complete	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:14:51	Quick Start Guide Harness	HAR-FL0001	A1	The Completed Harness		Not Started	
Fog Lamp Harness	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:31:35	Quick Start Guide Harness	HAR-FL0001	A	Fog Lamp Harness		Not Started	

Fig 4.1 The Drawing Data is Split Down In to the Modules, Drawing, Schematic & Harness.

## 5. The Schematic Window

Take a moment to look at the different areas of the Schematic Window (**Fig 5.1**).

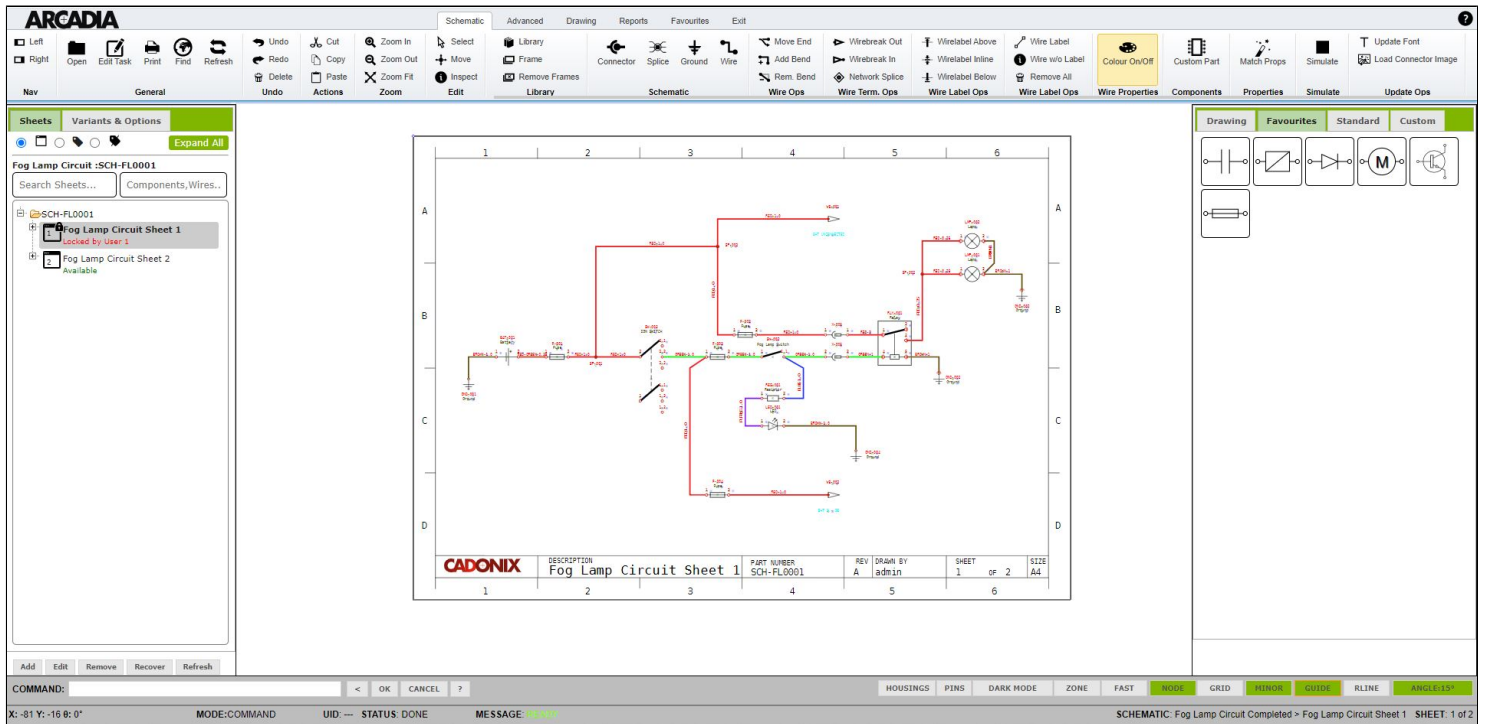
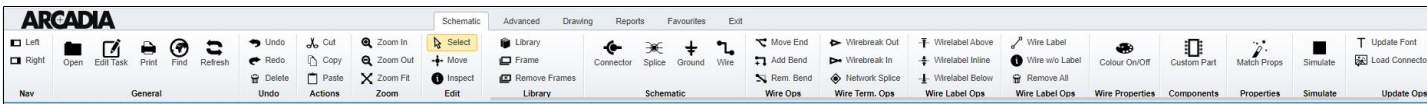
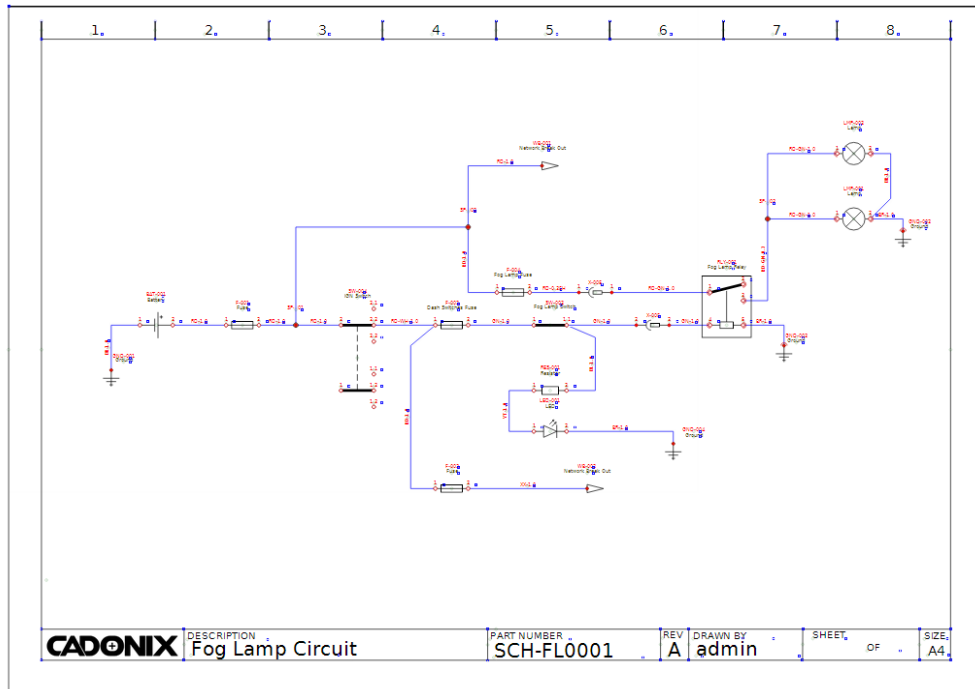


Fig 5.1 The Schematic Window

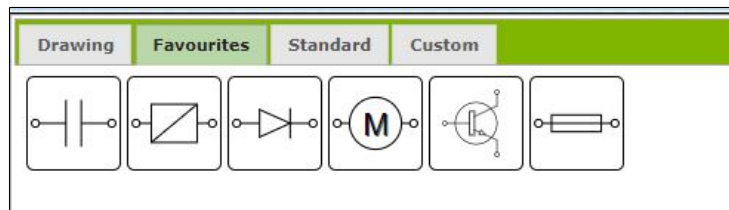
### 1 Toolbar



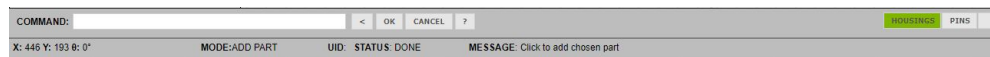
### 2 Worksheet



### 3 Library Navigator



### 4 Command, Message & Construction Toolbar



## 6. Mouse & Keyboard Commands



<b>D</b>	Dimension Shortcut
<b>F</b>	Insert Frame Shortcut
<b>J</b>	Arc Shortcut
<b>K</b>	Insert Leader
<b>L</b>	Line Shortcut
<b>M</b>	Move
<b>O</b>	Open Library
<b>Q</b>	Hide Project Navigator
<b>R</b>	Rectangle Shortcut
<b>T</b>	Text Shortcut
<b>U</b>	Curve Shortcut
<b>V</b>	Add Vertex to Polyline/Polygon
<b>W</b>	Hide Component Text Toggle
<b>X</b>	Pan
<b>Y</b>	Polygon Shortcut
<b>Z</b>	Zoom
<b>Del</b>	Delete Object
<b>Esc</b>	Quit current command
<b>Space</b>	Switch Activation

Advanced Keyboard shortcut details are available for additional functions by clicking the "?" link next to the command line at the bottom of the window.



**Left Button**

Used to select items

**Right Button**


Used to open menus and rotate items

**Centre Wheel / Button**

Used to Zoom when scrolled, Pan when pressed



## 7. The Library Navigator

You can show or hide the Library Navigator at any time by clicking on the **Toggle Navigator**  command - located top left hand corner of the screen.

Take a look at the **Library Navigator**, located to the right of the Schematic Window. The **Favourites** tab contains your frequently used components.

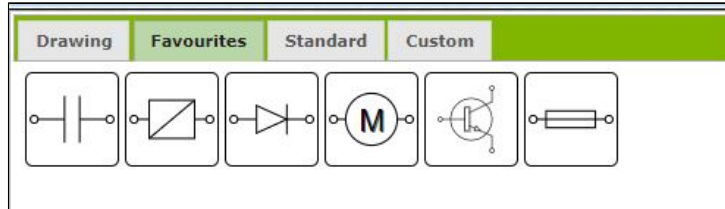


Fig 7.1 Favourites tab

Click on the **Standard** tab and expand the **standard** folder. You will now see a list of sub folders containing standard components in Arcadia.

Find the Fuse component using the **Search Bar**. Type '**Fuse**' and press Enter. Hover over the Fuse to see a preview. Hover over the Fuse to see a preview.

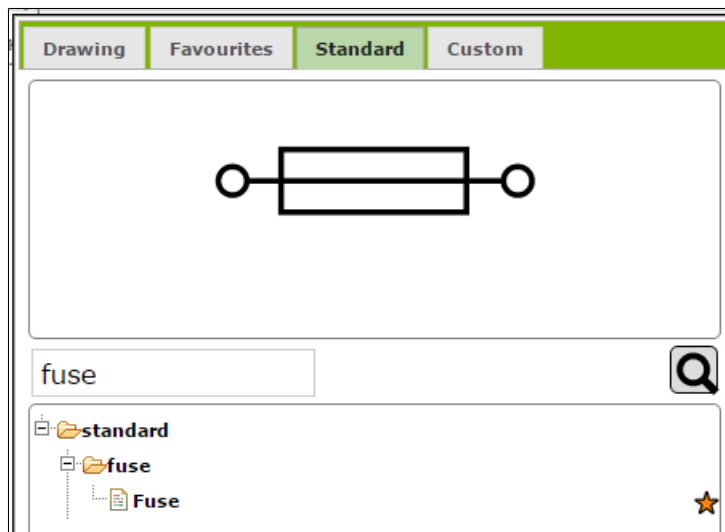


Fig 7.2 Libraries tab

Click on the **Star** icon to the right of the Fuse. The Fuse has now been added to your the list of your **Favourites**.

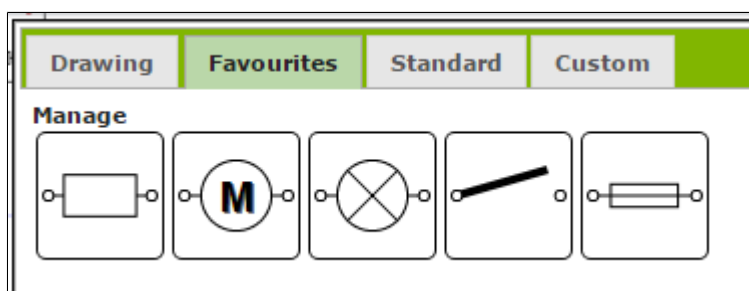
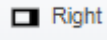


Fig 6.3 Favourites tab - This will differ based on the users choice. You can add your own favourites

---


You can show or hide the Library Navigator at any time by clicking on the **Toggle Navigator**  command - located top left hand corner of the screen.

## 8. Insert a Component

You are now going to add a **Fuse** to the circuit.

First locate the **Battery** on the circuit.

→ **Tip:** Use the **Zoom Fit**  command to see the whole worksheet.

You can also click on the **Find Component**  command to quickly find the Battery. Enter **Battery** into the search bar and click **Find**. Double click the Battery in the list and Arcadia will zoom to the battery and highlight the specific component.

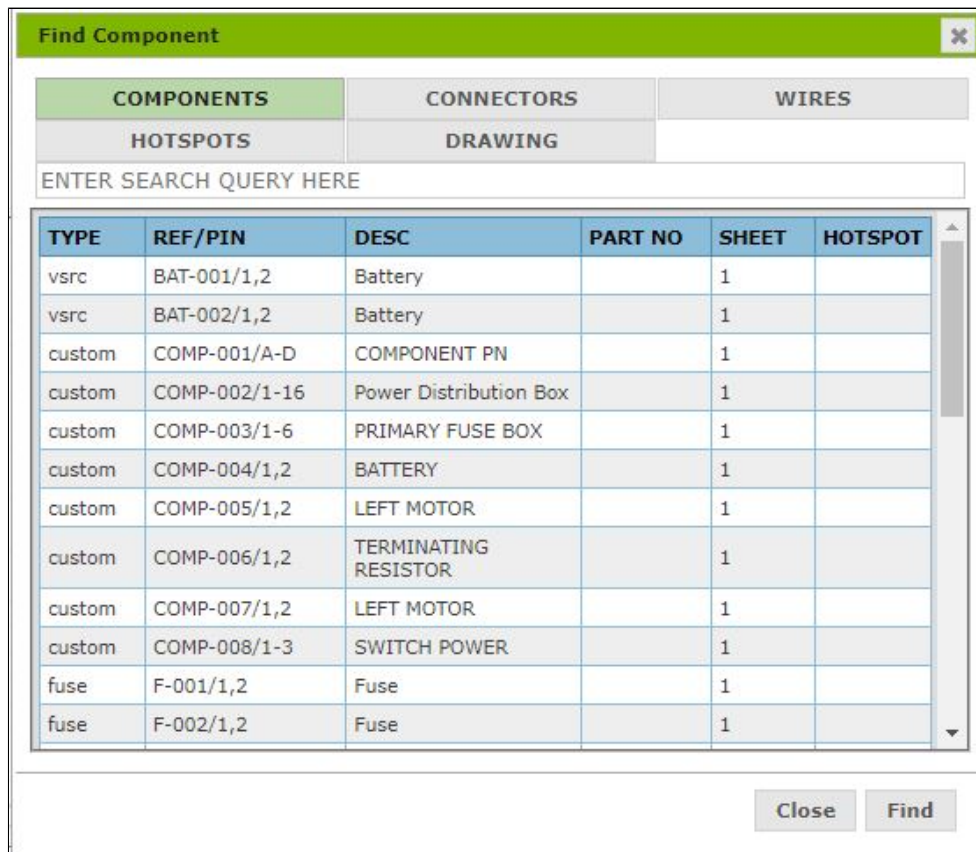


Fig 8.1 The Find Component Dialog box

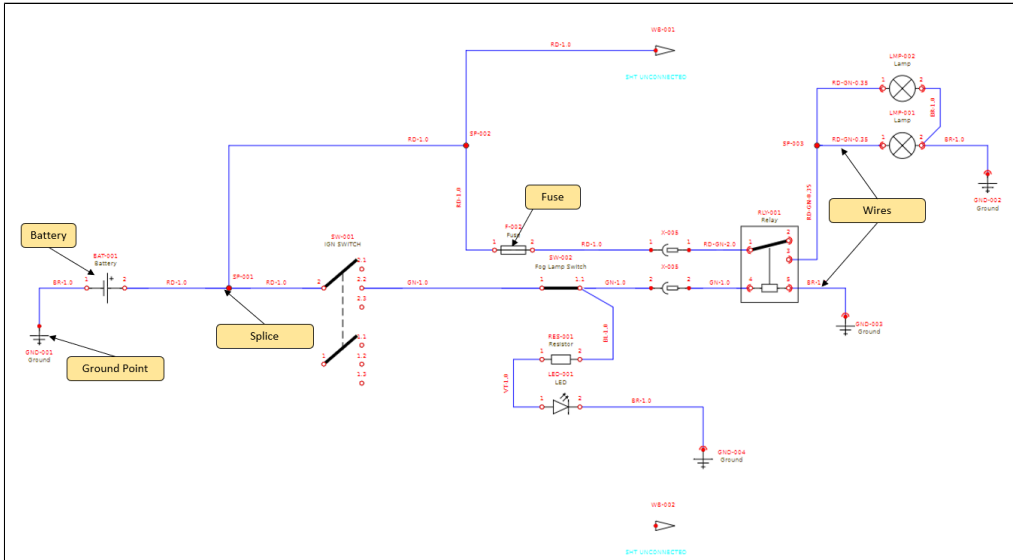


Fig 8.2 Fog Lamp Schematic

Using the **Library Navigator** on the right and under the **Favourites** tab, click on the **Fuse** and move your mouse onto the worksheet. You will see a ghosted image of the Fuse following your cursor. You will now place the fuse onto the wire as shown in **Fig 8.2** Place the Fuse on the wire by clicking once with the mouse. Arcadia will split the wire into two separate wires automatically.

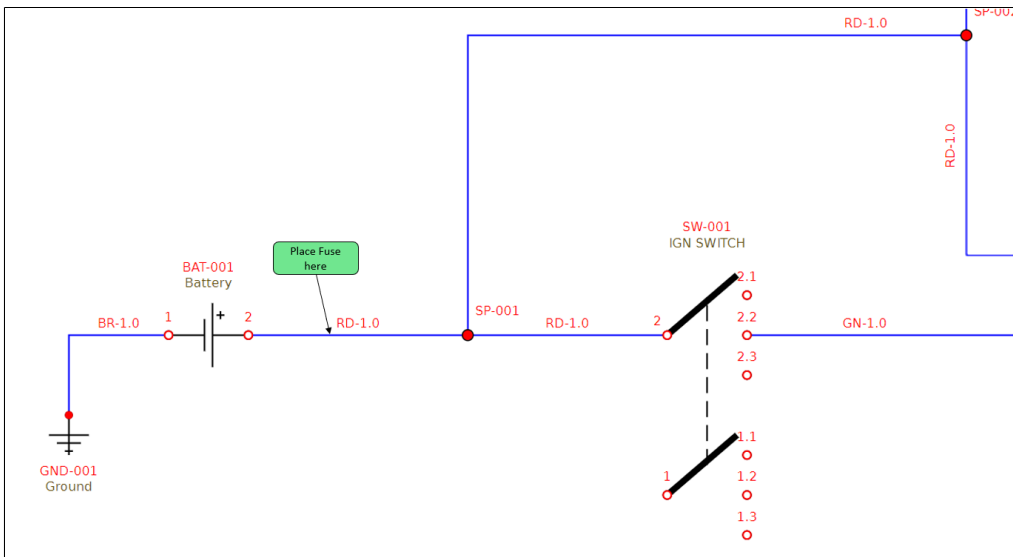


Fig 8.3 Fog Lamp Schematic with Fuse

Press **Esc** on your keyboard to quit adding another Fuse.

➔ **Note:** you can add components directly from the **Libraries** tab too.

You will now set the Fuse to be a 20 Amp Fuse.

**Right-click** on the Fuse you just added and select **Edit**. The **Edit Component** dialog appears.

Select the **Electrical** tab and set the **Fuse Rating** to be **20 Amps**.

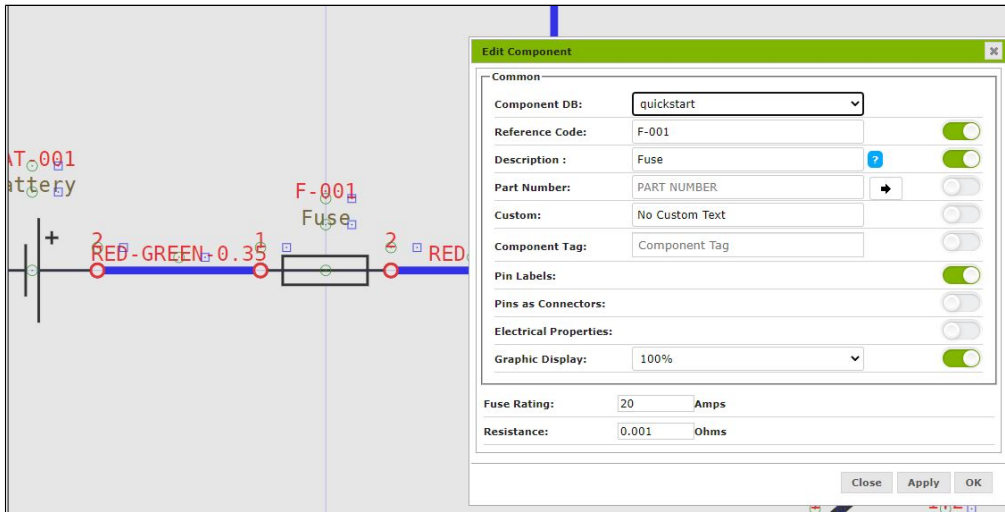


Fig 8.3.1 Edit Component dialog to set the Fuse Rating to 20 Amps

You can Add / Update the component details under the **Common** Tab I.E the Description, Part number, Reference number.

➔ **Note:** Right-clicking on an object on the worksheet will bring up the **Context Menu**.

Next add fuses F-003 & F-004 in the same way in the positions shown in **Fig 8.3**.

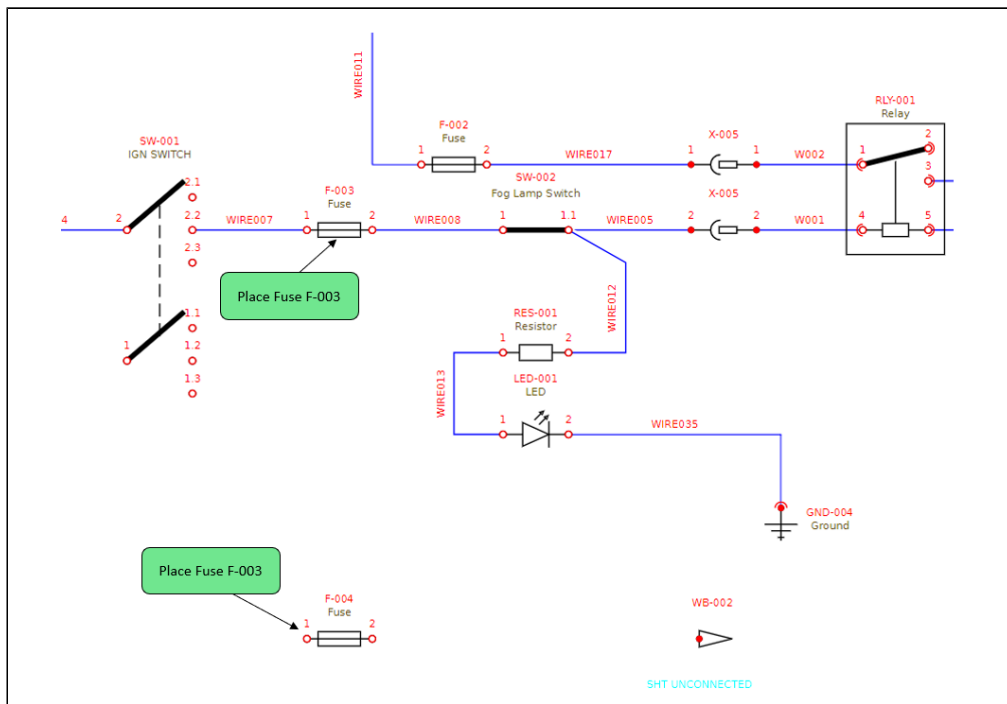
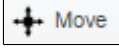


Fig 8.4 Placing F-003 & F-004

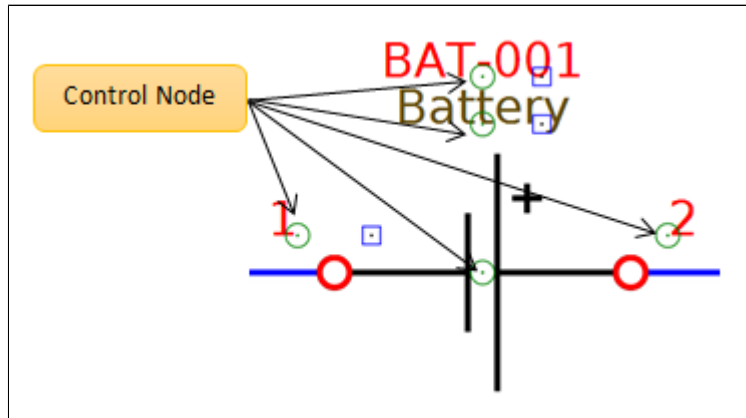
Set the rating of Fuses F-003 to 3 Amps and the Description to "Dash Switches Fuse", also set the rating of F-004 that you have just placed to 3 Amps.

\*Note Fuse F-004 has been placed unconnected in an open area for user later in the Quick start guide.

## 9. Move a Component

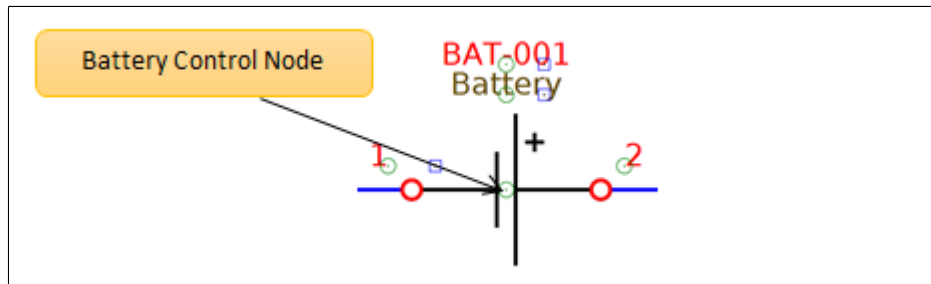
You can move components using the **Move** command in Arcadia. Click on the  icon (Located in Edit).

When moving an object you must click on a **Control Node**.



*Fig 9.1 Battery Control Nodes*

You will now move the Battery to the left. Click on the **Centre Control Node** for the Battery (Components will be highlighted red to give a clear indication they have been selected). Move the Battery to the left slightly and click to confirm your Move.



*Fig 9.2 Battery Centre Control Node*

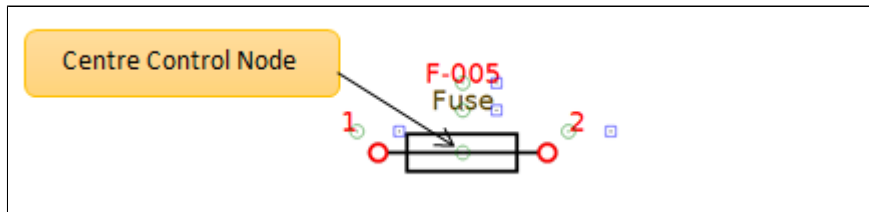
➔ **Note:** Moving the Centre Control Node for the Battery will move all corresponding texts. Control Nodes can be used to move Component text & Pin text individually.

➔ **Tip:** You can turn off Control Nodes by pressing **SHIFT + N** on your keyboard. Or turn them back on by pressing **SHIFT + N** once more.


Press the **Esc** key to exit the Move command.


## 10. Delete a Component

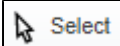
Use the Favourites tab to add another Fuse on to your worksheet. Place the Fuse into some free space.



*Fig 10.1 Fuse Centre Control Node*

You can delete components using the mouse right click Delete command in Arcadia. Or Click on the  Delete icon and click on the Centre Control Node of the Fuse you just placed down.

→ **Note:** You can use the **Del** key to delete components too. You can undo an action by clicking on the  Undo icon.

\*Note all selected Main (Top) menu commands will remain active until another is used or the ESC key is pressed (Returns the Selection to the **Select**  Select command.

## 11. Insert a Wire

You will now add a Wire linking the two Fuses **F-003** and **F-004**.

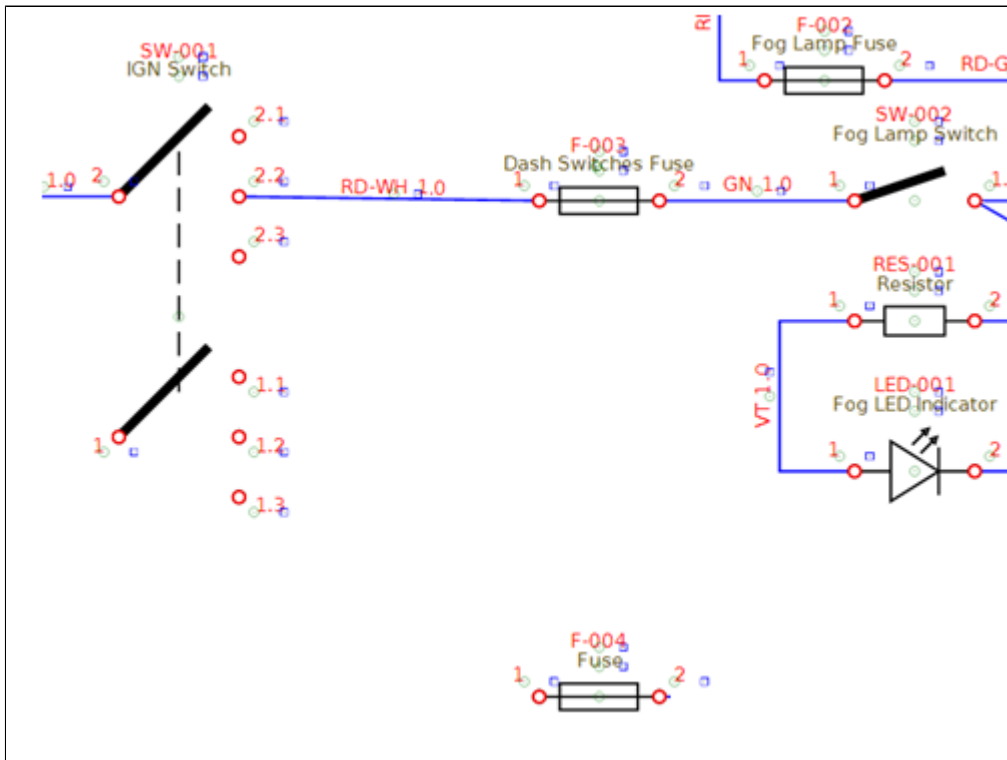


Fig 11.1 Fog Lamp Schematic



Click on the  icon. Begin by clicking on **Pin 1** of the **Dash Switches Fuse**. As you draw out your wire, click each time you wish to change direction.

Finish by clicking on **Pin 1** of **Fuse F-004**. When the wire connection is complete the **Edit Wire** options menu will appear just Click OK to continue, Your wire will now be completed.

Next add a wire from **Pin 2** of **Fuse F-004** to the **Wirebreak WB-002** in the same way, OK'ing the **Edit Wire** options again.

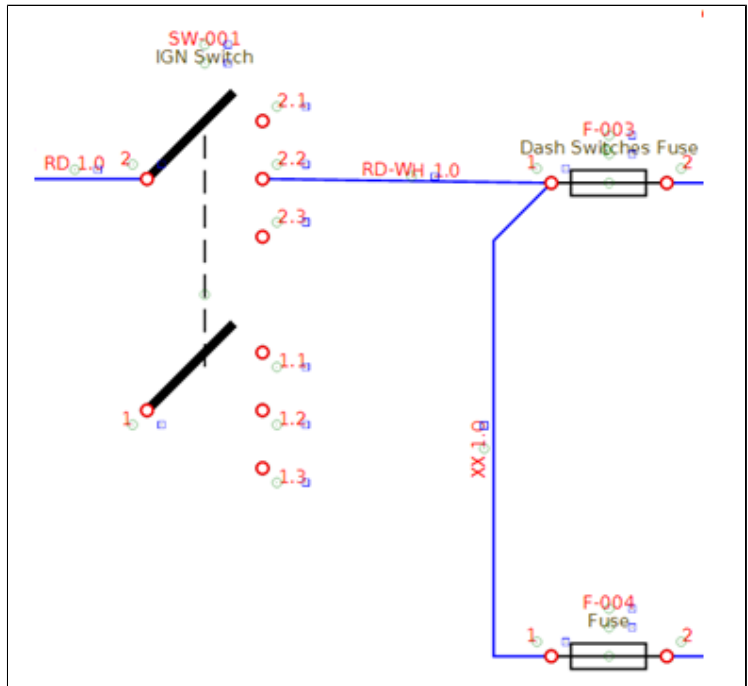



Fig 11.2 Fog Lamp Schematic with Wire Added

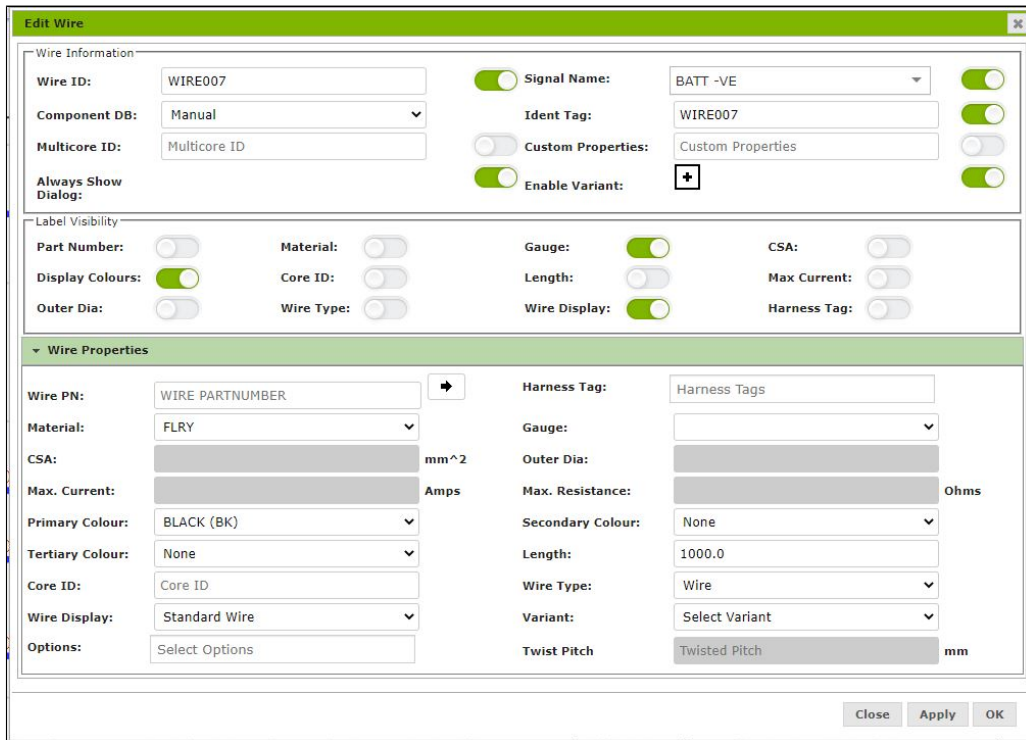
## 12. Inspect the Wire Properties

You can set property values in Arcadia such as Wire Label, Wire Name, Colour, Cross Sectional Area (CSA), Part Number (PN) and more.

You will now set the colour of the wire you have just added to be Red.

To do so, click on the  icon. Now click on the wire you added previously. This will bring up the **Edit Wire** dialog.

Set the **Primary Colour** to **RD**, Then **Apply** and **OK** the changes



Wire Information	
Wire ID:	WIRE007
Component DB:	Manual
Multicore ID:	Multicore ID
Always Show Dialog:	<input type="checkbox"/>
Signal Name:	BATT -VE
Ident Tag:	WIRE007
Custom Properties:	Custom Properties
Enable Variant:	+

Label Visibility			
Part Number:	<input type="checkbox"/>	Material:	<input type="checkbox"/>
Display Colours:	<input checked="" type="checkbox"/>	Core ID:	<input type="checkbox"/>
Outer Dia:	<input type="checkbox"/>	Wire Type:	<input type="checkbox"/>
Gauge:	<input checked="" type="checkbox"/>	CSA:	<input type="checkbox"/>
Length:	<input type="checkbox"/>	Max Current:	<input type="checkbox"/>
Wire Display:	<input checked="" type="checkbox"/>	Harness Tag:	<input type="checkbox"/>

Wire Properties	
Wire PN:	WIRE PARTNUMBER
Material:	FLRY
CSA:	mm <sup>2</sup>
Max. Current:	Amps
Primary Colour:	BLACK (BK)
Tertiary Colour:	None
Core ID:	Core ID
Wire Display:	Standard Wire
Options:	Select Options
Harness Tag:	Harness Tags
Gauge:	
Outer Dia:	
Max. Resistance:	Ohms
Secondary Colour:	None
Length:	1000.0
Wire Type:	Wire
Variant:	Select Variant
Twist Pitch:	Twisted Pitch mm

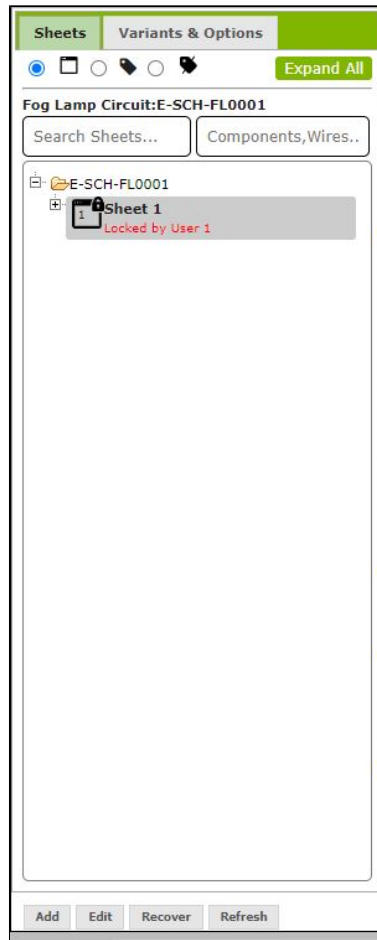
Fig 12.1 Edit Wire Dialog - Please note that Wire ID can be different to what is shown in this image

➔ **Note:** Double clicking on a wire will also bring up its properties. Wires have a default length of 1000.00mm

Press **Esc** or click on the  icon to exit the Inspection command.

### 13. Add a Drawing Frame

You will now continue the Fog Lamp circuit onto a new sheet. To do so, use the **Project Navigator** on the left hand side (*Fig 13.1*)



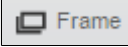
*Fig 13.1 Left Menu pane*

Create a new sheet called **Fog Lamp Circuit Sheet 2**. (Add Sheet button under **Navigator, Sheets**)

General Properties	Revision	Custom Properties
Custom Level:	Select Custom Level	
Description:	Fog Lamp Circuit Sheet 2	
Drawn By:	Sucheta Gavhane	
Drawn Date:	2023-02-07	
Part Number:	E-SCH-FL0001	
Status:	In Progress	
Editable Status:	Editable	

Cancel OK

Fig 13.2 Add Sheet window

You will need to add a frame to the worksheet. Click on the  icon. On the right hand side the **Add Frame** dialog appears (Fig 13.3). Select **A4 Landscape Standard** from the drop down list. **Click Submit**. If the frame you have added is placed in different location, please click Zoom Fit icon from the tool bar.

Drawing Favourites Standard Custom

**SELECT YOUR FRAME:**

▼ Frames

X Base: 0mm

Y Base: 0mm

X Elements: 1

Y Elements: 1

Frame: A4 Landscape Sta ▼

▼ Advanced

X Spacing: 0

Y Spacing: 0

Submit Cancel

*Fig 13.3 Frame options menu*

Your frame will be added to the worksheet.

## 14. Wire Breaks

Wire breaks are used to electrically connect two segments of the same wire which may be represented on different pages of a circuit.

From earlier, notice we had a wire break named **WB-002** on Sheet 1.

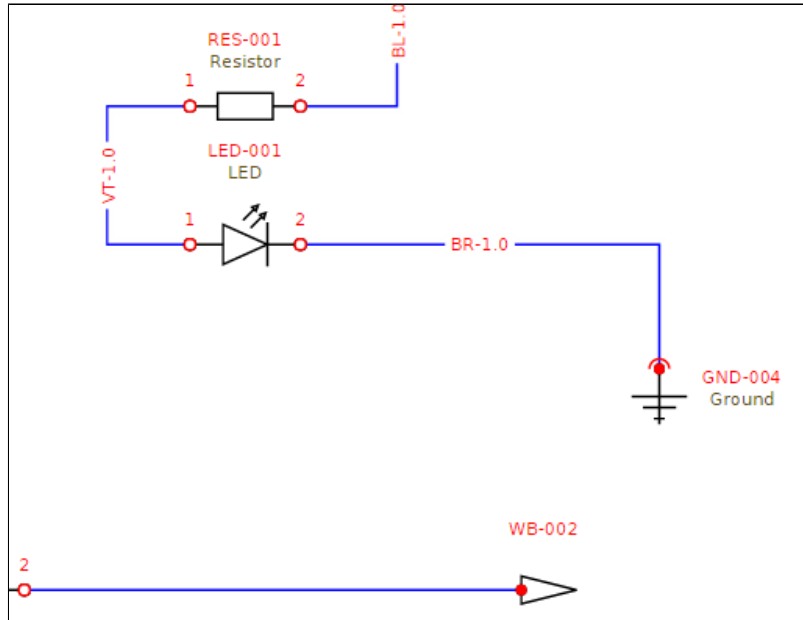


Fig 14.1 Wire Break WB-002

You will now continue this part of the circuit on Sheet 2 of your circuit.

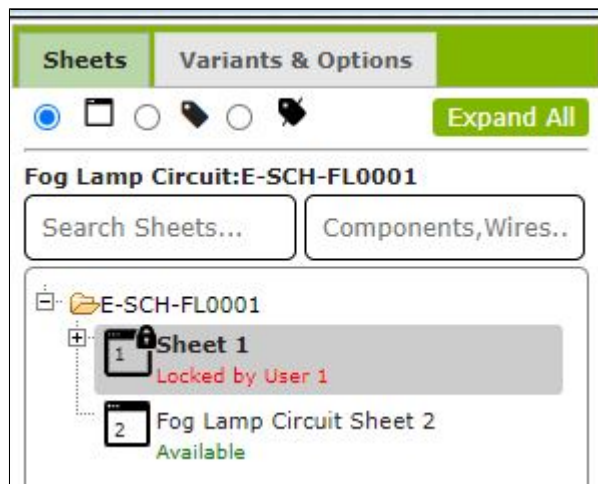



Fig 14.2 Select Sheet 2 from the Project Browser

Click on **Fog Lamp Circuit Sheet 2**. Now use the **Wire Break In**  command. Place the Wire Break onto the blank sheet.

Click **Select**  then **Right-click** on the Wire Break (Note clicking the Red wire connection node will bring up a different menu) and select **Networks**.

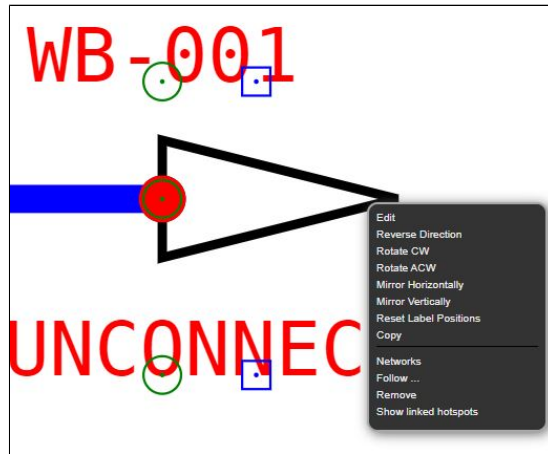


Fig 14.3 Wire Break Right Click Menu

Change the **Network** to **WB-002** so that the Wire Break is linked to the existing Wire Break WB-002 and click OK.

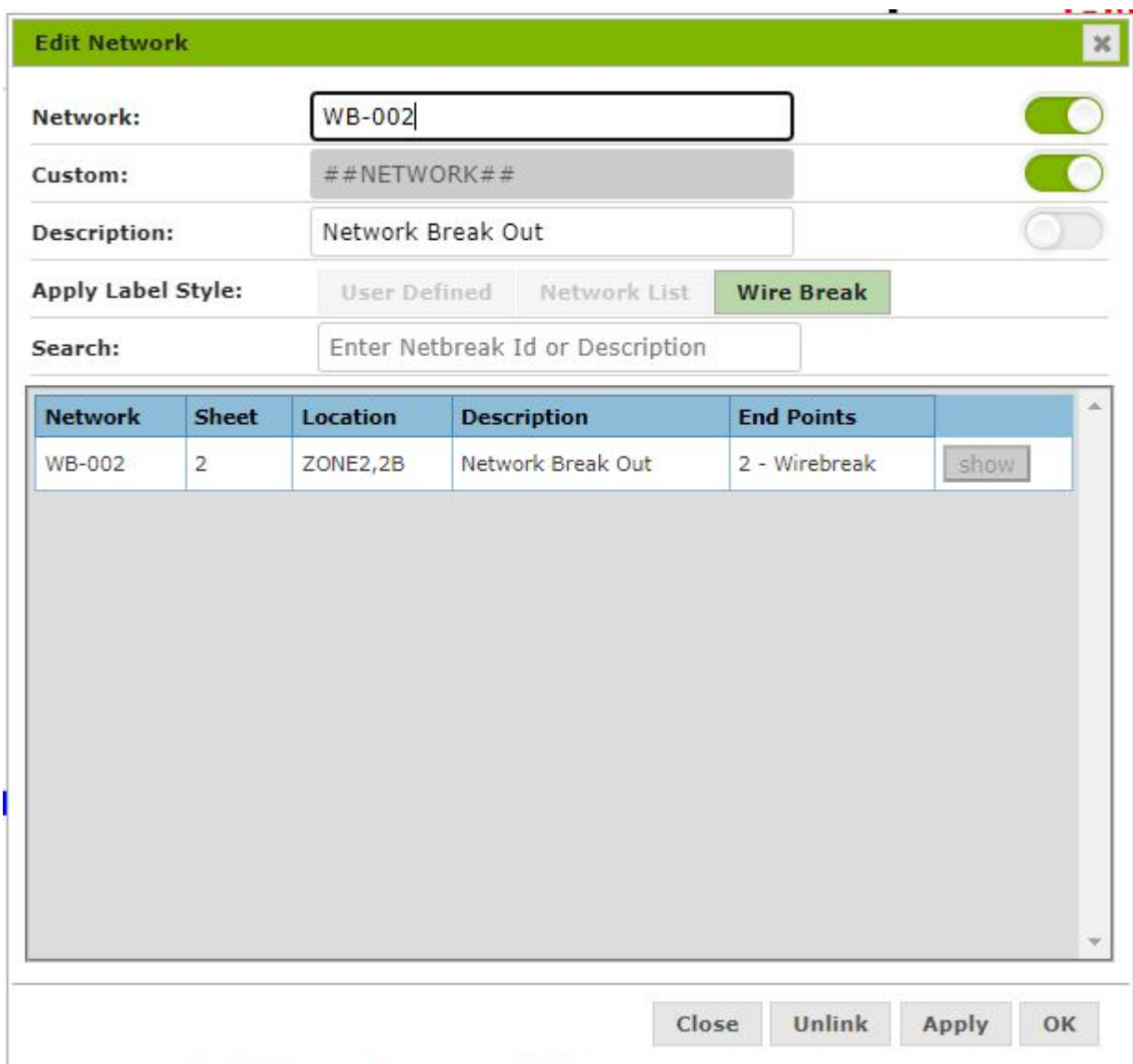


Fig 14.4 Wire Break Properties

By using the **Favourites** tab, you will now add the **1-pole 1-contact Switch** and **Lamp** onto the worksheet as shown in Fig 14.5.



To place the Ground, use the

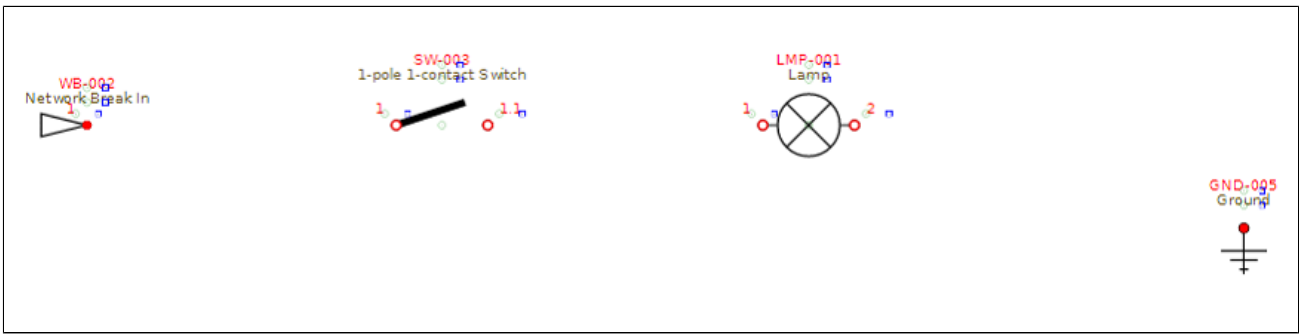


Fig 14.5 Sheet 2 Components

→ **Tip:** Turn guide on to better align your components. Select the **GUIDE** command located at the bottom right of the screen.



Add wires to your circuit using the

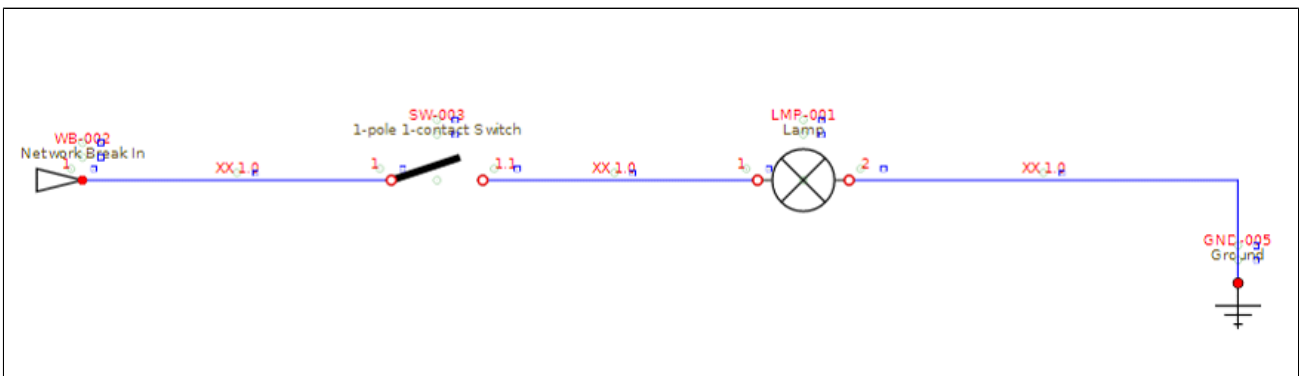


Fig 14.6 Sheet 2 Completion

## 15. Simulation

Arcadia has a powerful built in simulation capability. Fuse, wire and component ratings can be checked, as well as voltage drops and electrical backfeed detection.

You will now run simulation on the Fog Lamp circuit to check for potential problems.



Open Sheet 1 of the schematic. By default Simulation is off, clicking on the **Simulate** icon once will run Simulation for the current worksheet.

You can pull in the switches interactively on the worksheet. Double click on the **IGN Switch (SW-001)** and the **Fog Lamp Switch (SW-002)**

Note: To use switches ensure the **Select** button is selected.

The circuit will update and provide visual feedback.

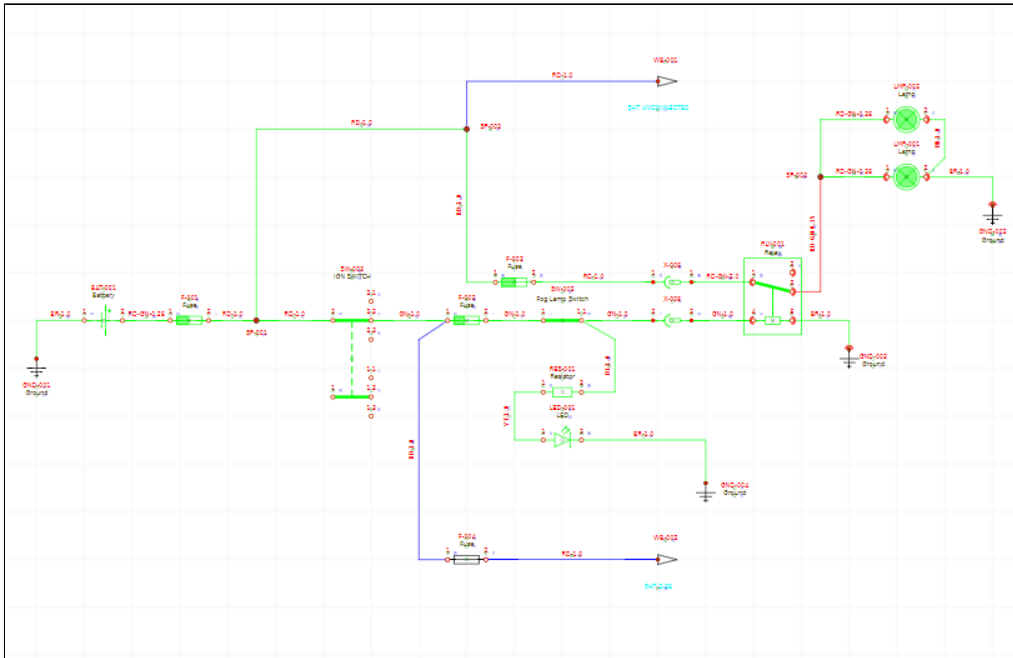


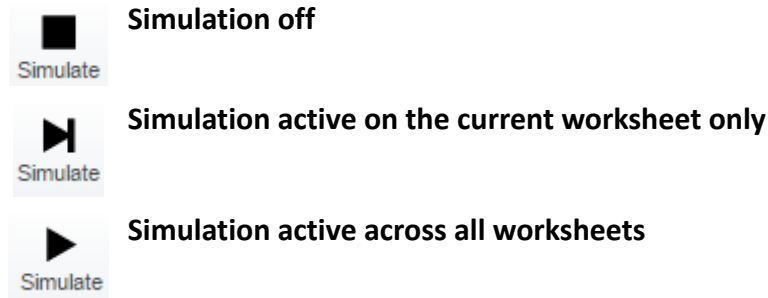
Fig 15.1 Simulation Active

➔ **Tip:** When pulling in the switch, hover over the switch first until it is highlighted in purple, and then double click the switch.

The simulation states are represented by the following colours:

- |               |  |
|---------------|--|
| <b>Green</b>  | <b>Component working to specification / wire conducting within specification</b> |
| <b>Blue</b>   | <b>Component running under specification / wire not conducting</b>               |
| <b>Red</b>    | <b>Component failed / wire conducting out of specification</b>                   |
| <b>Orange</b> | <b>Warning – working close to specification</b>                                  |
| <b>Black</b>  | <b>Component off or not able to simulate</b>                                     |

Clicking once more will run Simulation on the entire circuit across all sheets. The simulation modes are represented by the following icons, clicking on them will cycle through the simulation modes:



## SIMULATION LOG

The simulation log is modified to show the actual reference code of the component in question with a hyperlink to the drawing to help the users to debug effectively.

The simulation log can be generated while running the simulation. The “simlog” command is used to generate the log report of the circuit operations.

Simulation Details
✕

Input Summary	Results	Warnings & Issues	
W005	LMP-002/2	LMP-001/2	1.85e-7
W008	LMP-001/2	GND-002/1	1.85e-7
WIRE023	F-003/1	F-004/1	0.0185
WIRE002	SP-001/1	SP-002/1	0.0185
WIRE004	SP-001/1	SW-003/1	0.0185
WIRE005	SW-002/2	X-005/1	0.0185
W001	X-005/2	RLY-001/1	1.85e-7
W004	GND-003/1	RLY-001/2	1.85e-7
W002	X-005/2	RLY-001/3	1.85e-7

**\*Models:**

```
.model ledRGYAGaAs d (IS=93.1P RS=42M N=4.61 BV=4 IBV=10U CJO=2.97P VJ=.75 M=.333 TT=4.32U)
```

\*Subcircuits :

**\* battery model**

```
.SUBCKT BATTERY1 1 2
V1 1 3 DC 12
R1 3 2 0.012
.ENDS
```

**\* Fuse Model**

Close

Fig 15.2 Simulation Log

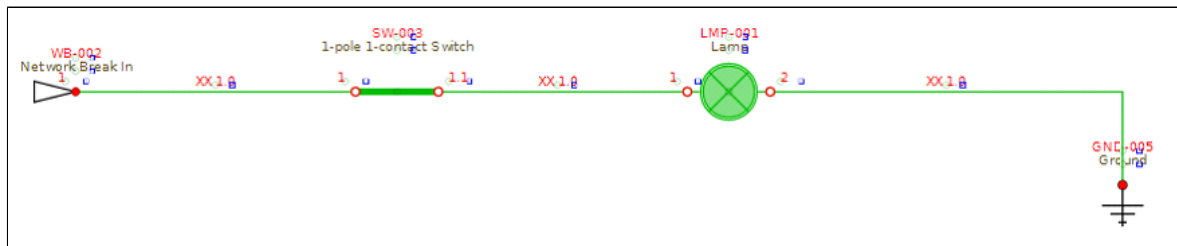
## 16. Diagnose & Fix

Run the simulation so that it is active across all sheets.

Select **Sheet 2**, and pull in the switch you added earlier (Ensure that that you have run the simulation across both



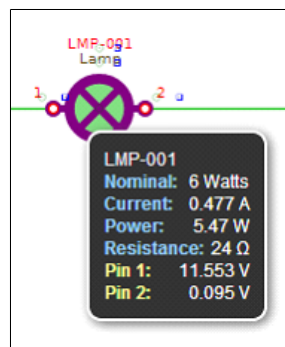
the sheets by pressing the Simulate again ).



*Fig 16.1 Simulation Active Across All Sheets*

The circuit updates and the Lamp is now on.

Using Arcadia it is possible to inspect wire and component values that are being simulated. Hover your mouse over the Lamp.



*Fig 16.2 Simulation Tooltip*

A tool-tip appears including the simulated data for the Lamp.

Navigate back to **Sheet 1**.

There is a problem with the wire highlighted in Red. Hover your mouse over the wire to view the tooltip.

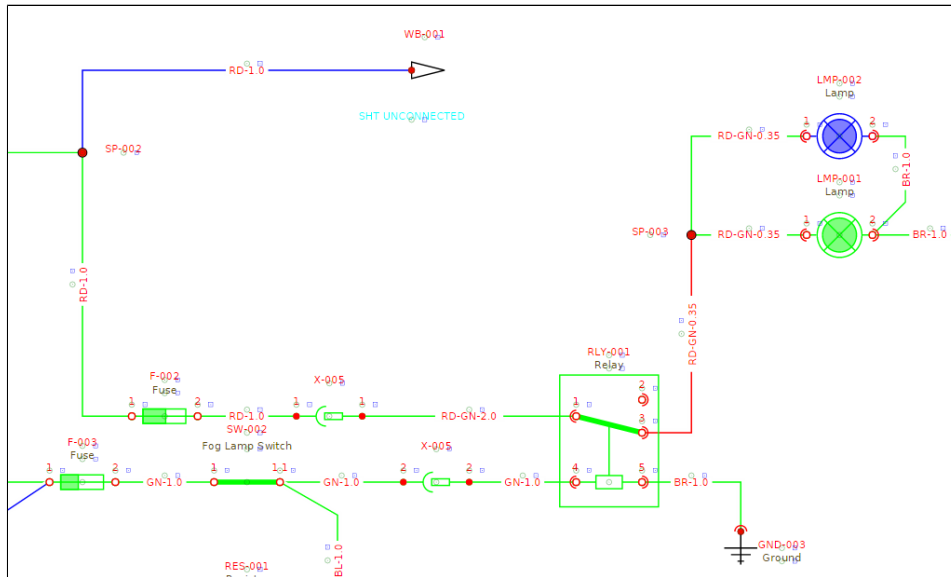


Fig 16.3 Problem Wire

The wire (W003) has a maximum rated current of 7A however the actual current through the wire is higher. The simulation has correctly detected an excess current in the wire.

Notice that the two Fog Lamps are highlighted in blue. This is because they are performing under specification. They are rated at 55W each, however the actual power they are producing is less than 40W in the current simulation.

➔ **Tip:** Arcadia provides visual feedback for fuse loads. The percentage for each fuse load is shown by the amount the fuse has been filled in by.

You will now fix the problem wire.

Whilst simulation is running, double click on the red highlighted wire (W003). Increase the **Gauge** of the wire to **2.0** from the drop down. Click **OK**.

Take note of the updated simulation.

## 17. View Wire Colours

You can display the actual wire colours on the worksheet for your circuit.

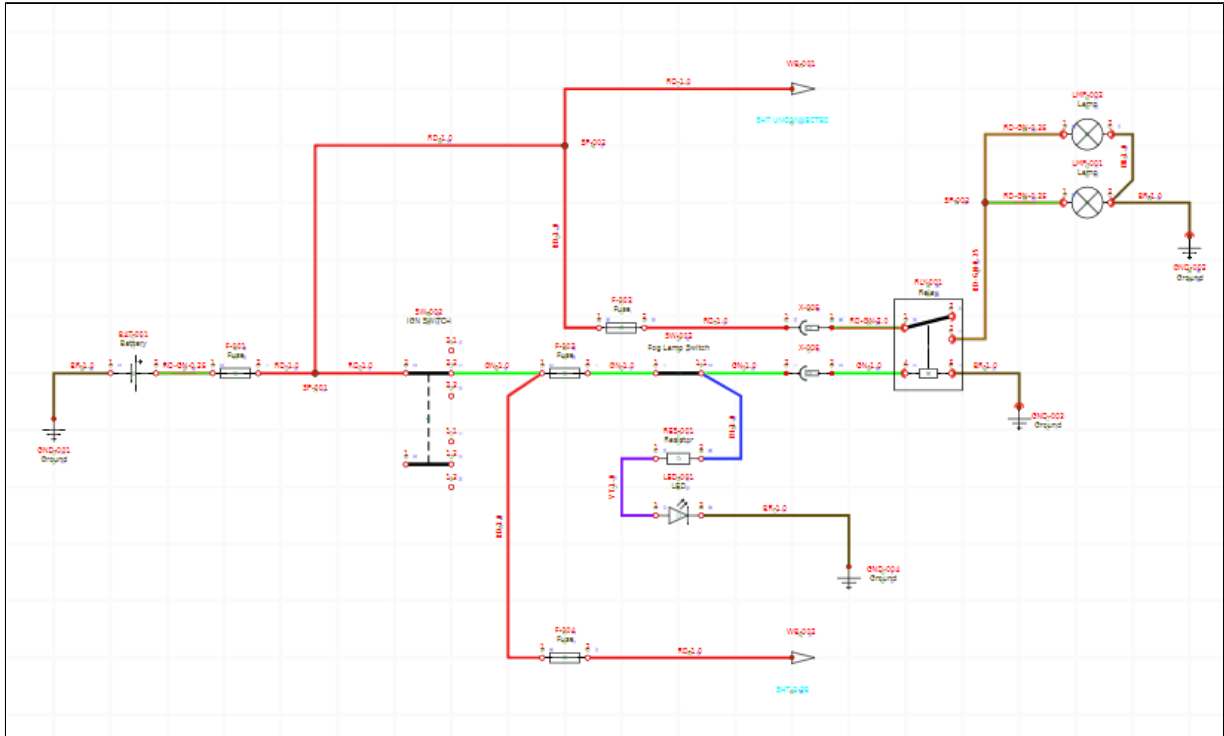
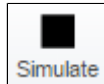


Fig 17.1 Wire Colours

You will need to first turn off all simulation



Now click on the **Wire Colours**  icon.

The circuit updates to show the actual wire colours used in the schematic.

➔ **Note:** By Default orange is also used to denote wires that have not had their colour set, this can be changed in the profile.

## 18. Find an Object on the Schematic



You can search for a component/wire by using the **Find** command on the toolbar

Click in the **Search Box** and press enter to list all Components. Alternatively you may enter a text you wish to search for first and press enter. The search result is listed as shown in Fig 18.1.

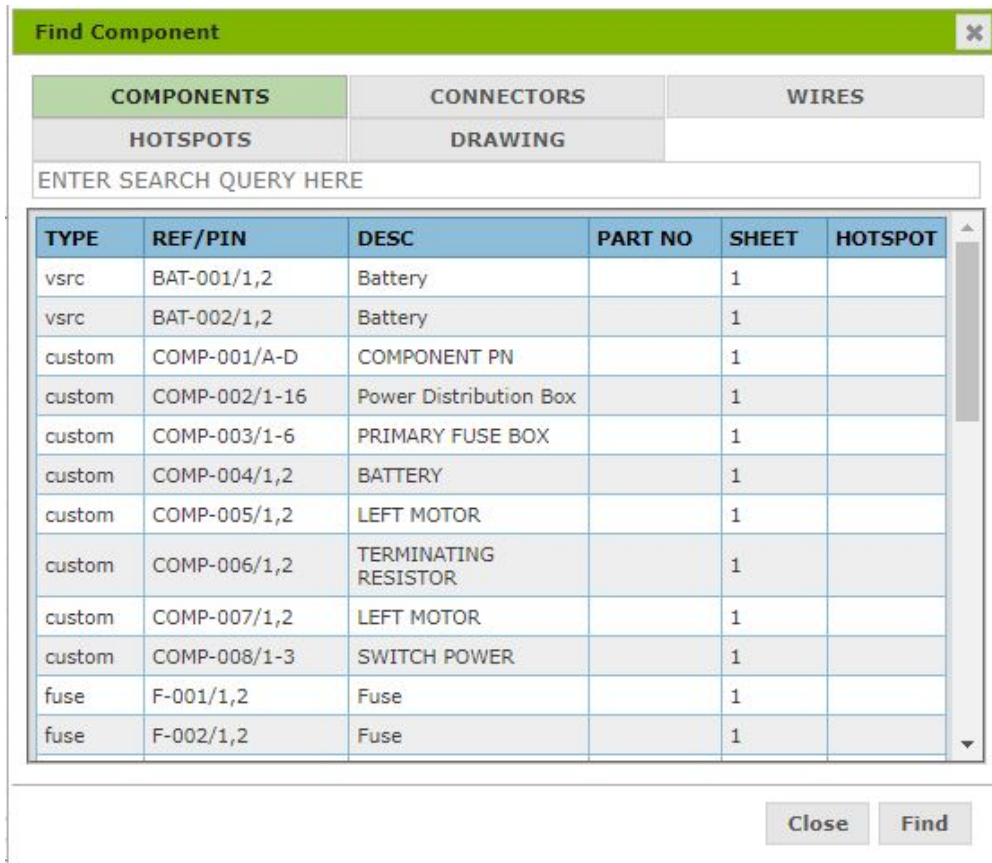


Fig 18.1 Find an Object

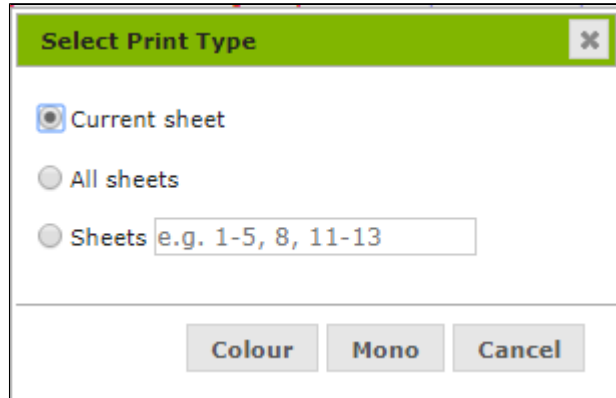
You can click on an item in the list. The corresponding component will highlight on the worksheet.

➔ **Tip:** You can search by **Wires**, **Components** and **Connectors**

## 19. Print the Schematic



The Print command allows the schematic to be printed in colour or monotone. This will generate a new window with your schematic available as a PDF or to be printed directly. You can also select which sheets you would like to print by range.




→ **Tip:** To print the actual wire colours, turn on Wire Colours first before clicking the Print command




→ **Tip:** To print only an area of the schematic pre-select the area before clicking the Print command by clicking and dragging over the area with the tool

## 21. Wire Labelling

We will now add a label to the one of the wires and set the label location.


To change the wire Label, Click the Inspect icon  Inspect and then the control node or wire you wish to label (Wire08)

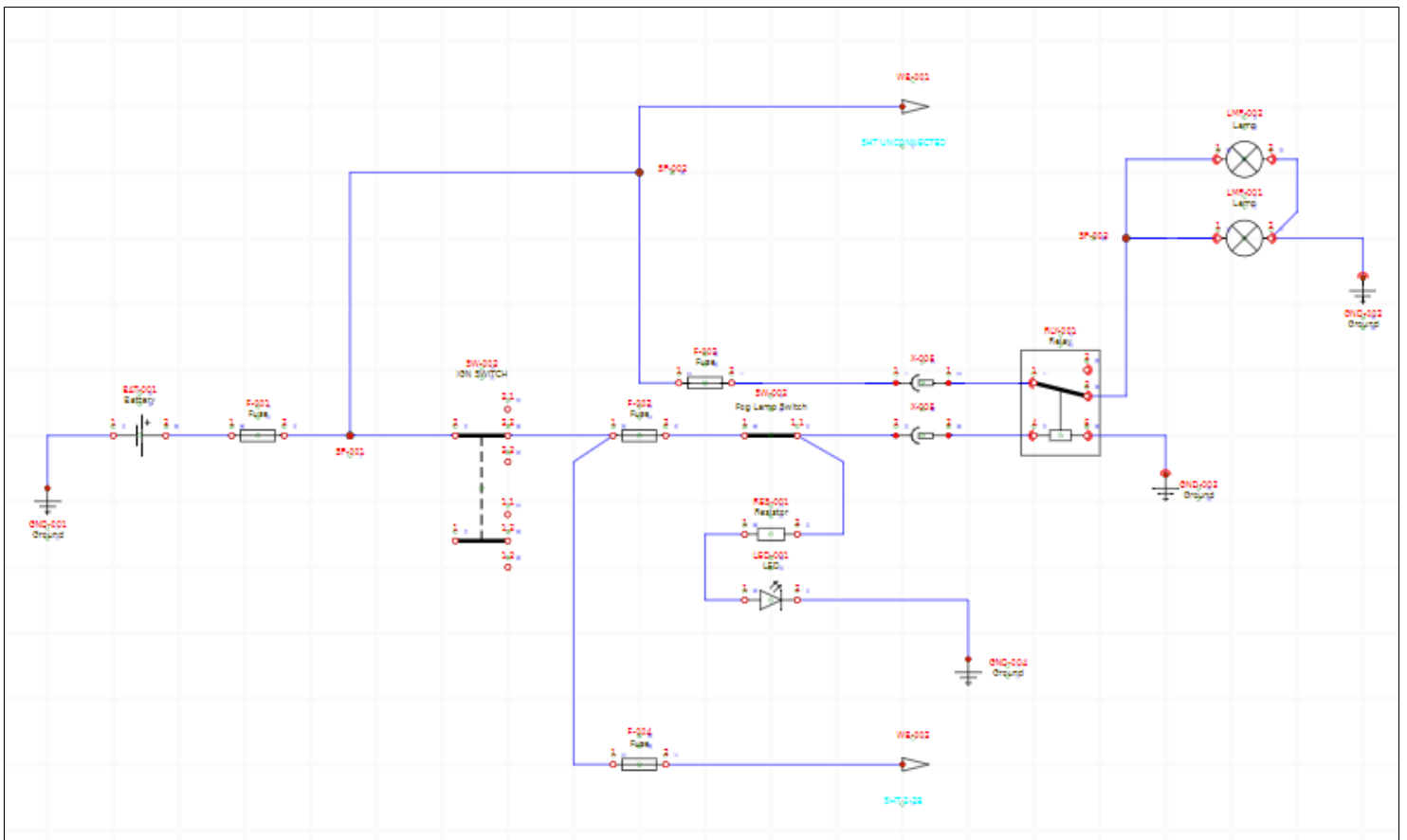
Under Wire ID change to FOG CIRCUIT 2.

You can change the wire label location of all wires in the project by clicking the wirelabel above , wirelabel inline  or wirelabel below  icons.

In addition to the wire label options above it is also possible to use the new "Wire Label Ops" tools:




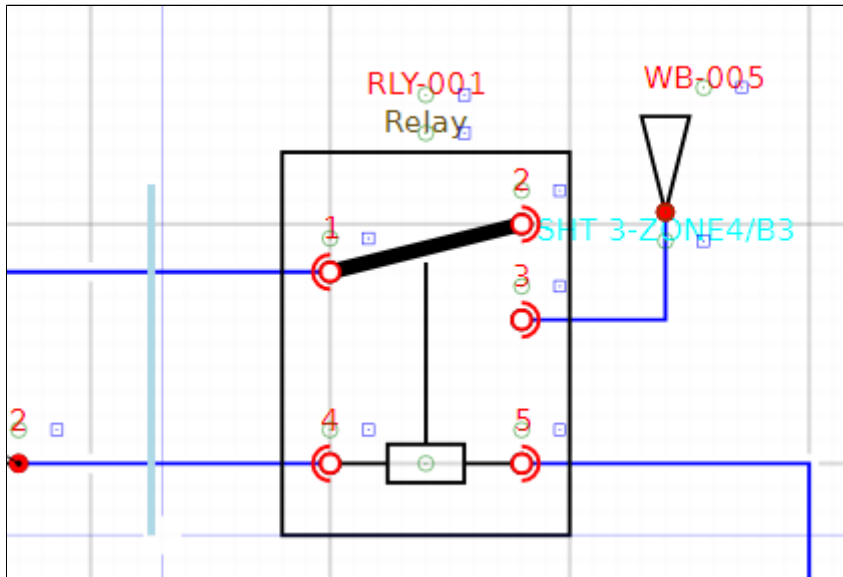
Clicking the  Remove All button will erase all existing wire labels as shown in the image below:



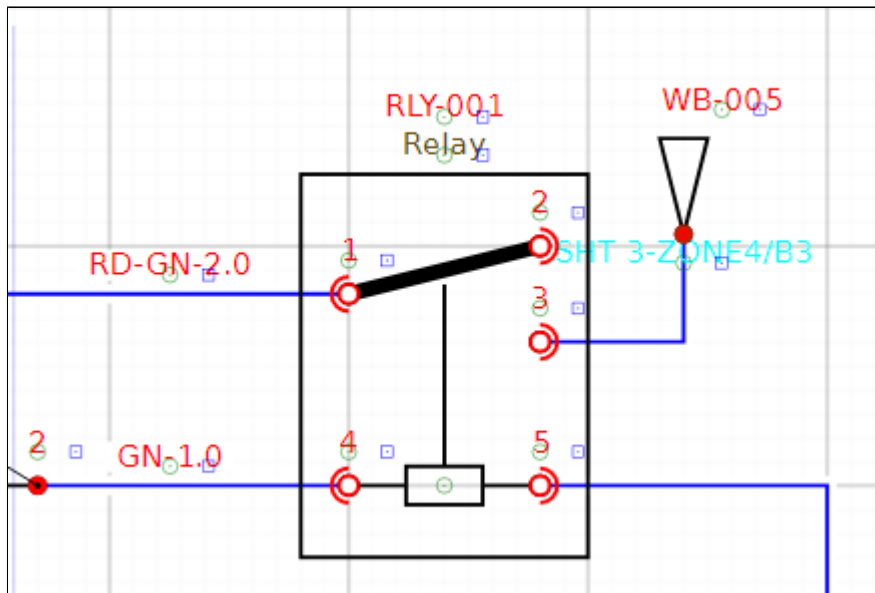
*Drawing a wire label line across intersecting wires*

Note: this function can also be used on a elected area if you do not want to remove all labels on the sheet.

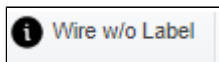
Using the  Wire Label tool drawing a horizontal line across the unlabeled wires will add labels neatly along the line as shown:

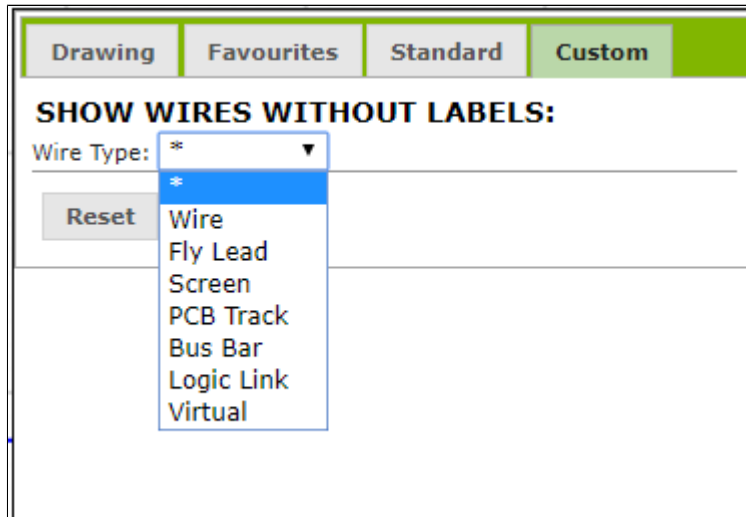


wire label line drawn alongside RLY-001

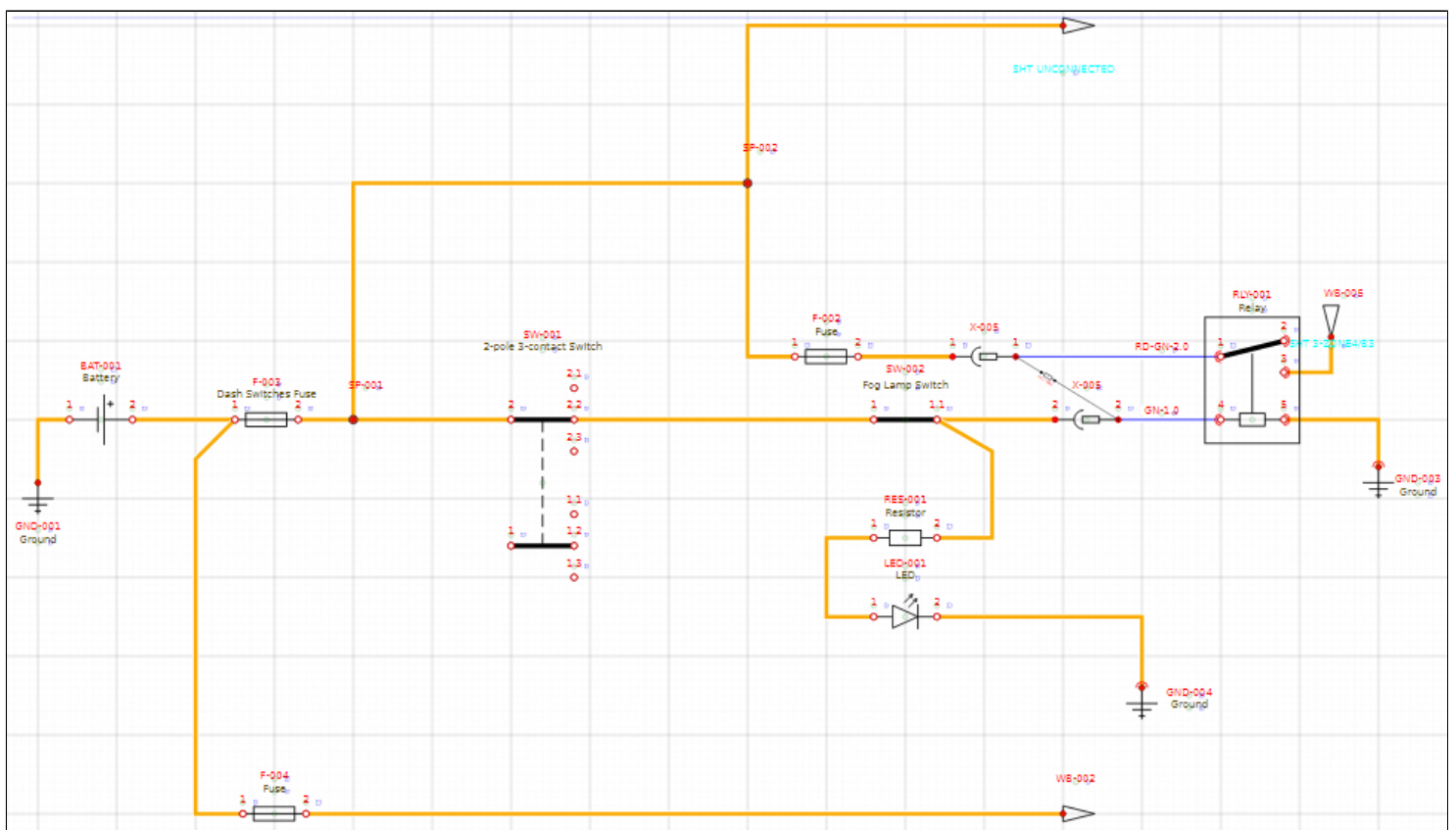


resulting wire labels

If you would like to be shown which wires you have not yet labelled click the  tool. A dialogue will allow you to filter certain types of wires if required such as PCB tracks and fly-leads. \* shows all items, click submit to display.



wires without labels options



Highlighted wires with no label

The unlabelled wires will highlight orange to identify which remain to be labelled on your current sheet

## 22. Discrete Component

We will next add a discrete component to the connector X-005 after SW-002 (Fig 22.1)

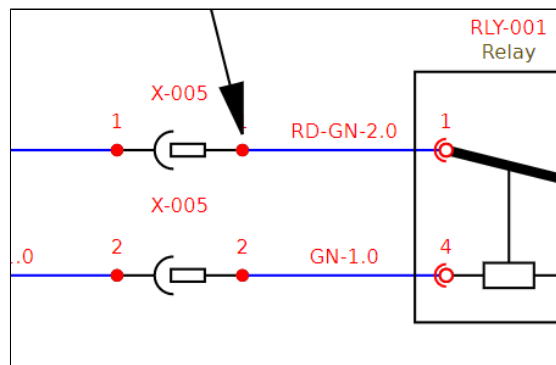



Fig 22.1 Connector X-005

Click the **Inspect**  icon and then the male control node arrowed in Fig 22.1. Select the **Discrete** tab then click Add Component, Set the options as detailed below in Fig 22.2 and click **OK** to apply them.

The screenshot shows the 'Edit Inline Pins' dialog box with the 'Discrete' tab selected. The 'Add Component' button is highlighted. The 'From Pin' is set to 1, 'Destination Type' is Pin, 'Dest. Pin/Wire' is 2, 'Discrete Type' is Resistor, 'Component DB' is Manual, 'Ref Code' is Auto, 'Discrete Part No.' is DISCRETE PART NUMBER, and 'Part Desc.' is Part Description.

Fig 22.2 Discrete component addition properties

A new discrete component has now been added between male pin 1 and 2 in Connector X-005 as shown in Fig 22.3

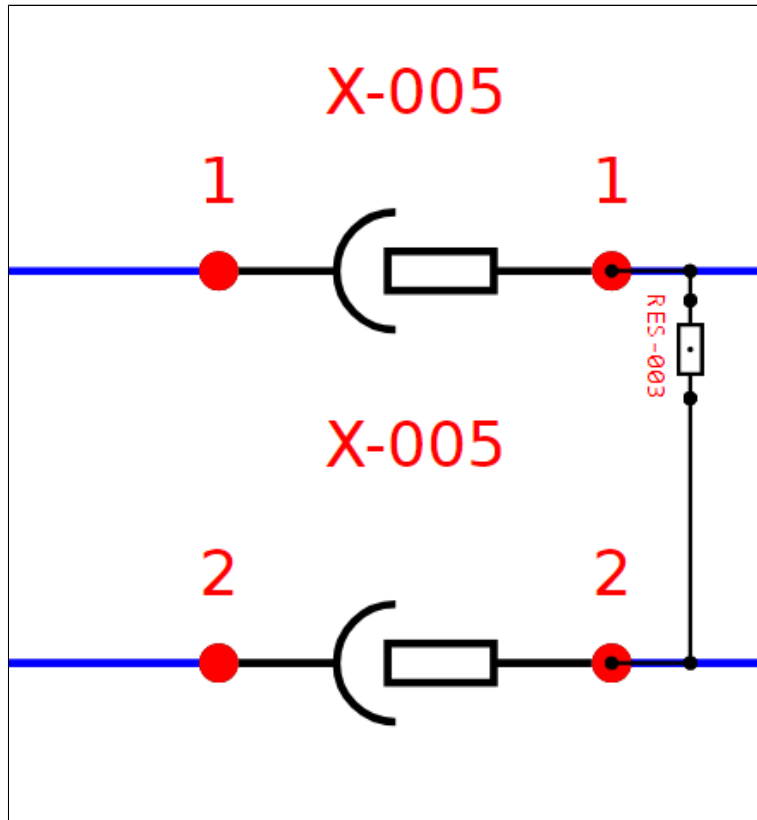



Fig 22.3 Resistor discrete component between pin 1 and 2 in connector X-005

Note\* Discrete components are intended to be added inside a back shell of the connector when manufactured as typical for military and motor-sport harnesses.

### 23. Add Wire Breaks to Split a Schematic

We will next add a Wire break to split the fog lamps to a 3rd Sheet.

Locate wire RD -GN 2.0 running from pin3 on relay RLY-001, Select the **Wire break in** tool  and right click the mouse to rotate the wire break to point vertically (Fig 23.1) left click to place the wire break

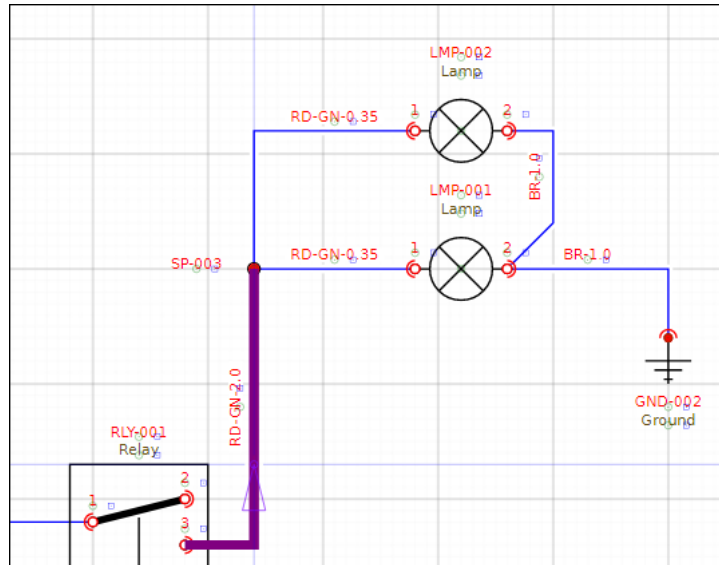





Fig 23.1 Wire break In on wire RD-GN 2.0

We will next split the schematic at the wire break and send the Fog lamp section to a new Sheet.

Click the Add Sheet icon  at the bottom of the Sheets pane on the left of the screen, under the Description type **Fog Lamp circuit Sheet 3**, and OK to add the sheet. Add a Frame to the new sheet by clicking the **Frame tool**  Select **A4 Landscape Standard** under the **Frame**: pull down and click **Submit** to complete. Switch back to **Fog Lamp Circuit Sheet 1** and Click the **Select tool**  and draw a box around the Lamps and GND - 002 selecting through the wire break (Fig 23.2)

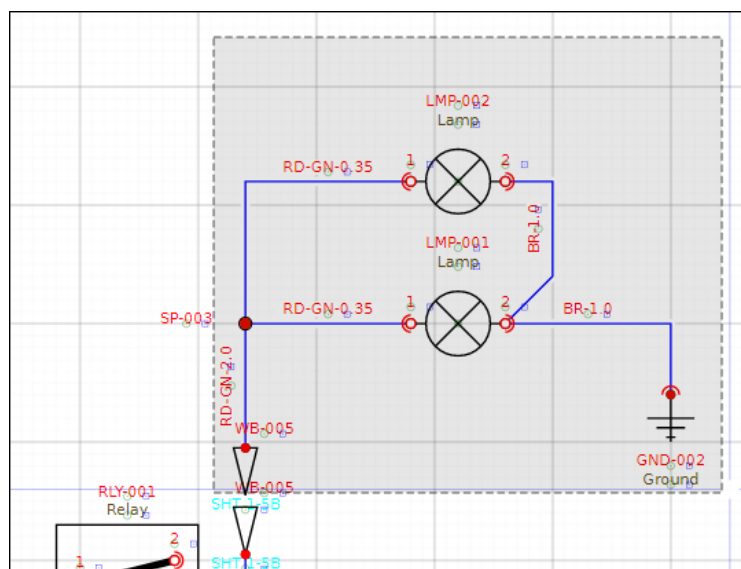


Fig 23.2 Selection of components after Wire break WB-005

Right click in the selected area to open the menu and click **Send to sheet** (Fig 23.3)

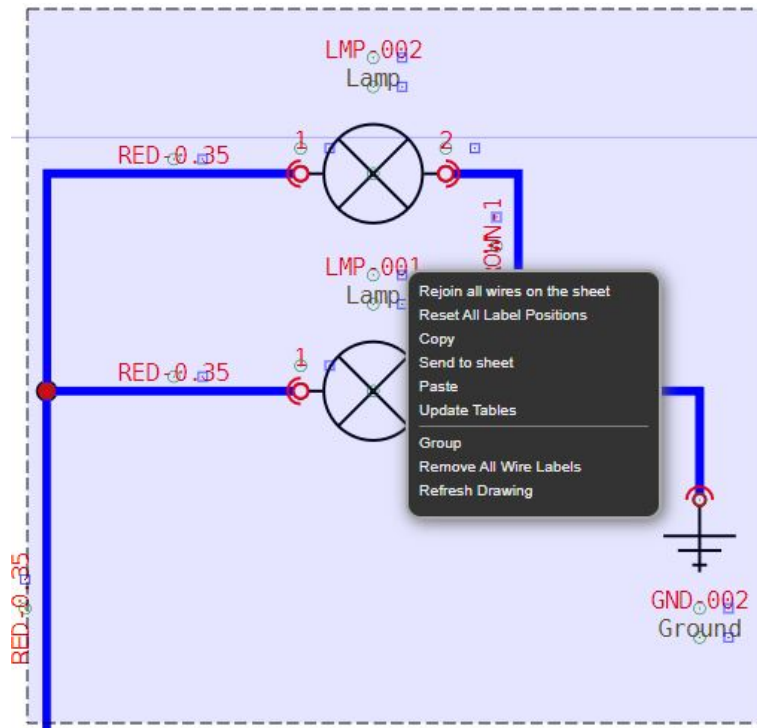



Fig 23.3 Right click "selected area" options menu

Select **Fog Lamp Circuit Sheet 3** and OK.

Sheet 3 will open and the selected area will be attached to your cursor, then you will then need to place your components within the frame on your Sheet, left click at the desired location.

To label our new wire break click the Inspect  tool on the control node of the new wire break, the Edit Network dialogue box will appear set the **Network:** to **ON** (Fig 23.4) this adds the WB-005 label name to you schematic for easy identification.

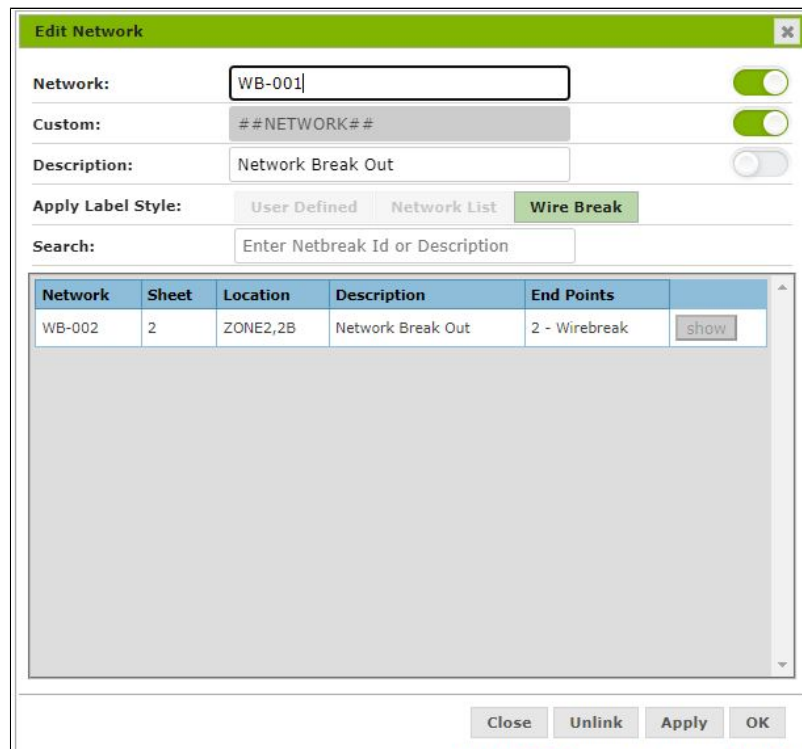


Fig 23.4 Wire break In properties

Your components will be linked via the wire break and labeled to the Sheet and Grid Zone location (Fig 23.5 Sheet 1, Fig 23.6 Sheet 3)

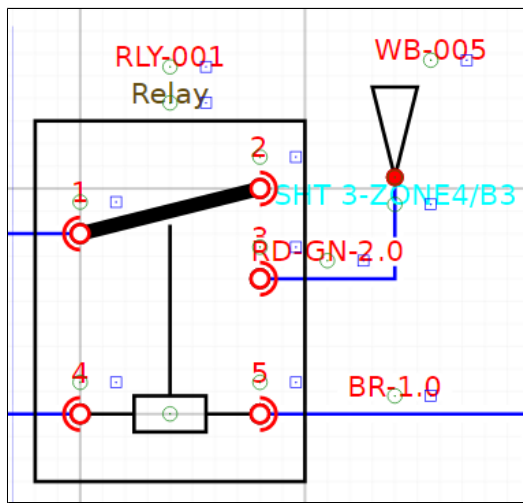


Fig 23.5 Wire break components in sheet 1 linked

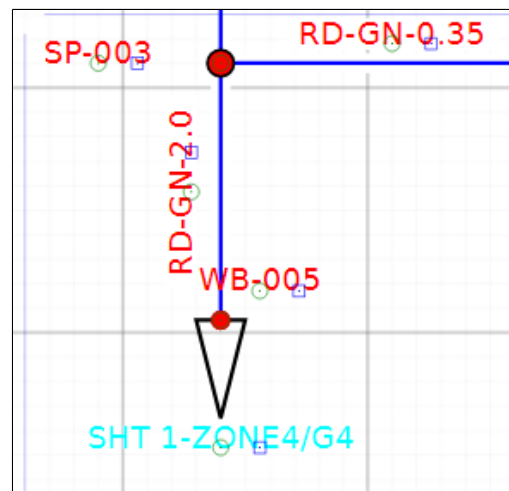
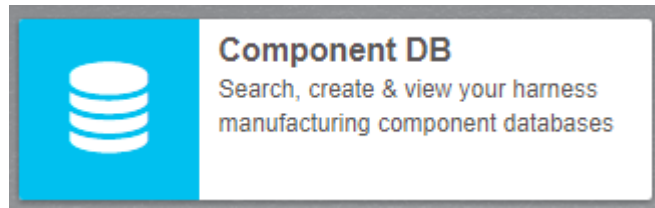


Fig 23.6 Wire break components in sheet 2 linked

---

## Component Database

Using the Arcadia Component Database toolset, you will add a new Terminal and Connectors for use in the Fog Lamp Harness.



*Link to Component Database Manager in the Home Menu*

## 25. Add a Terminal

We will next create a new terminal entry in the database.

From the **Home** menu (**Fig 25.1**) select **Component DB** (**Fig 25.2**) next select **Quickstart** (**Fig 25.3**)

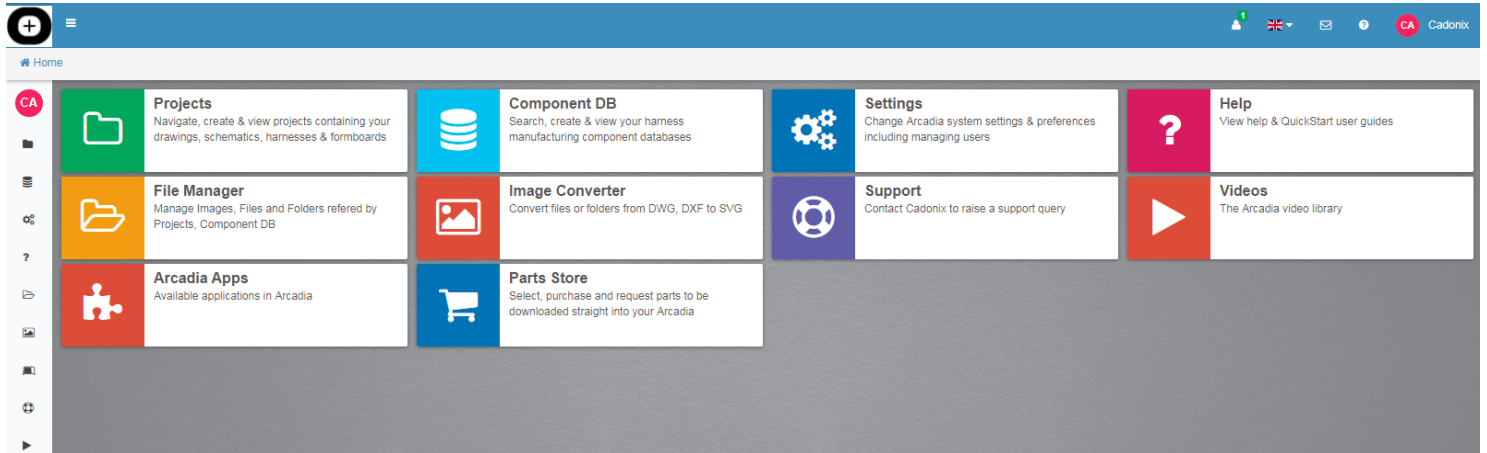


Fig.25.1 Home Menu

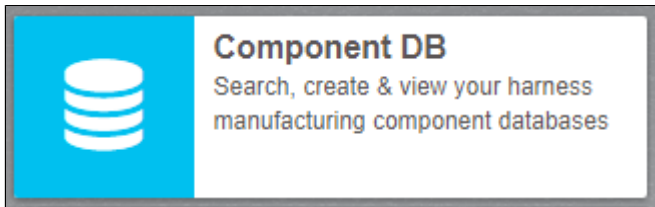


Fig 25.2. Component Database Select button

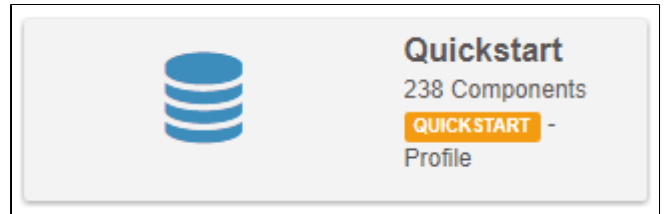


Fig 25.3. Quickstart Select button

Click the **Terminals** 🔍 Terminals (73) tab then **Add New Component**

**+ Add New Component**

Add the following details to the Add new connector screen (**Fig 25.4**)

Description : M6 Ring Terminal

Colour : YELLOW

Family : Type "Ring Terminal"

Status : IN REVIEW

Part Number : 0-7654321-9

Type : Supplier

Company: Type "CADONIX"

Fig 25.4. Add new Terminal properties screen

Click [Create New Component](#) to add it to your database.

Next select the CSA range of the terminals from 1 to 3.0mm<sup>2</sup>, you may upload images for the connector at this point or we can create a "Generated View" for the connector using the View generator described separately in section 25.

To save these changes click the [Update Component](#) link at the bottom of the page.

## 26. Add a Connector

Next we will add 2 new connectors to our Component database, A square with 6 cavities and a Circular with 5 cavities.

From the **Home** menu (**Fig 26.1**) select **Component DB** (**Fig 26.2**) next select **Quickstart** (**Fig 26.3**)

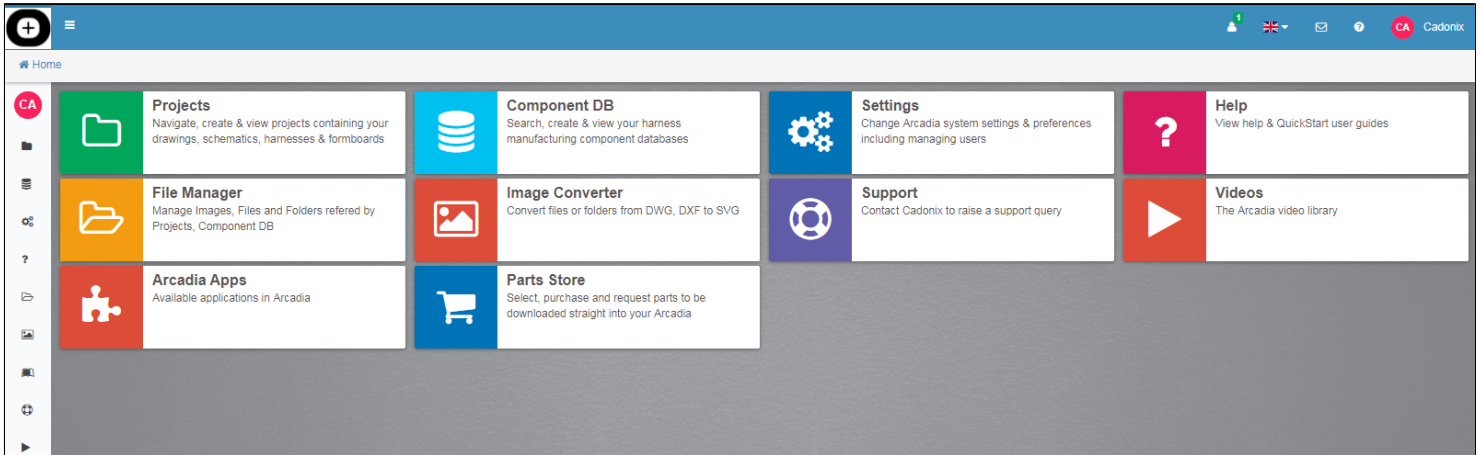


Fig 26.1 Home Menu

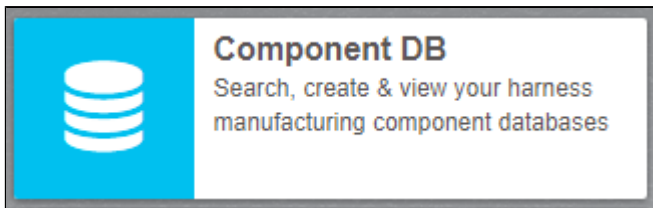


Fig 26.2. Component Database Select button

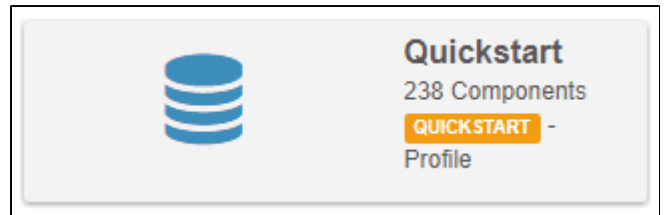




Fig 26.3. Quickstart Select button

Click the **Connectors**  tab then **Add New Component** 

Add the following details to the Add new connector screen (**Fig 26.4**)

Description : Square 6 Cavities

Colour : Blue

Family : Blank

Status : IN REVIEW

Part Number : 0-1234567-9

Type : Supplier

Company CADONIX

**Add New Connector**

**Details**

Description: Enter Description  
 Family: Select/Enter Family  
 Type Code:   
 Proprietary:   
 Part Type:   
 Colour: None  
 Status: IN REVIEW  
 Material Code:   
 Usage: NOT SET  
 Part Category:   
 + Add Row

**Additional References**

Part Number	Type	Company	Options
0-12345678-9	Supplier	CADONIX	CAD

**BOM Details**

Price: 0.00  
 Weight: 0  
 Measure: EACH  
 Currency: GBP  
 Units: gm  
 Bill Type: INCLUDED (MATERIAL)

Reset Create New Component

Fig 26.4. Add new Connector properties screen

Click **Create New Component** to add it to your database.

Next we will add a Circular with 5 cavities.

From the **Home** menu select **Component DB (Fig 26.5)** next select **Quickstart (Fig 26.6)**

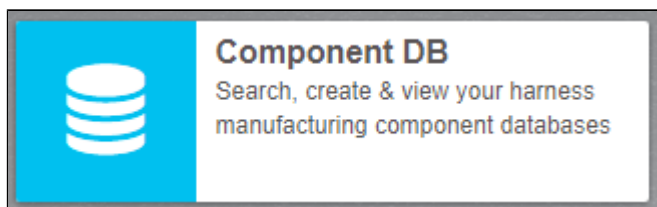


Fig 26.5. Component Database Select button

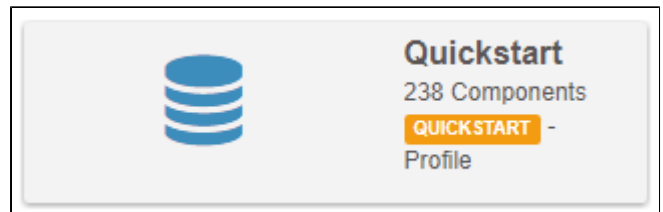


Fig 26.6. Quickstart Select button

Click the **Connectors** **Connectors (34)** tab then **Add New Component**

Add the following details to the Add new connector screen (**Fig 26.7**)

Description : Circular 5 Cavities

Colour : GRAY

Family : Blank

Status : IN REVIEW

Part Number : 0-12345678-9

Type : Supplier

Company CADONIX

Fig 26.7. Add new Connector properties screen

Click **Create New Component** to add it to your database.

In the new Connector record configure some details for the connector for example the number of cavities it has, in this example we are setting 5.

You may upload images for the wire connector at this point or we can create a "Generated View" for the connector using the View generator described separately in the next section.

Select terminals to fit your connector, they can be selected from the existing examples available in the Quickstart database:

Fig 26.8. Select Load From Database

The screenshot shows a window titled "Available Terminals" with a table of terminal specifications. The table has columns for Part Number, Description, Wire C, Wire IN, Finish, Family, Gender, and Supplier. There are also buttons for "Export Data" and "Export Custom Fields" at the top left, and a search bar at the top right. The table contains five rows of data. At the bottom, there is a pagination control showing "Showing 1 to 25 of 425 rows" and a "25 rows per page" dropdown. A navigation bar at the bottom right shows page numbers 1, 2, 3, 4, 5, and 17.

Part Number	Description	Wire C	Wire IN	Finish	Family	Gender	Supplier	Options
1062-16-0166	FEM TERM_STAMPED SIZE 16_0.75-2.0mm2	0.75-2.0	1.9-3.55	TIN-NICKEL	STAMPED SIZE 16	FEMALE	DEUTSCH	+
8240-4140-03	HSQ .250 FEMALE CONTACT	0.50-2.00	1.70-2.80	TIN	HSQ .250	FEMALE	SUMITOMO	+
0-0170258-1	.250 SERIES RECEP CONT. W/O LATCH	2.00-3.00	2.90-3.80	UNPLATED	LANCED .250 SERIES	FEMALE	AMP	+
0-0152404-3	SHUR PLUG RECEP TERMINAL	0.50-1.50	2.30-3.10	NONE	SHUR-PLUG	FEMALE	AMP	+
0-0150555-0	SHUR PLUG RECEPTACLE	1.25-3.00	3.10-4.30	NONE	SHUR-PLUG	FEMALE	AMP	+

Fig 26.9. Select Load From Database

Select a range of terminals that you would like to link to the connector with differing wire ranges and select

 Update Component

to save your new connector.

## 27. Create a Base Point & Automatic Add-On Generator

The next step is to add a Base Point for one the new connectors.

From the **Home** menu select **Component DB** (Fig 27.1) next select **Quickstart** (Fig 27.2)



Fig 27.1. Component Database Select button

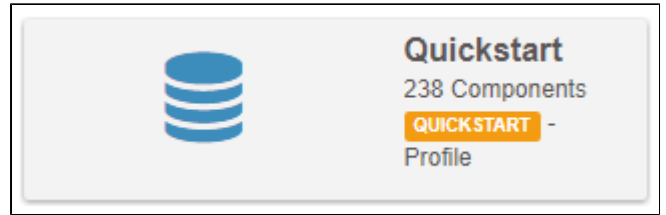
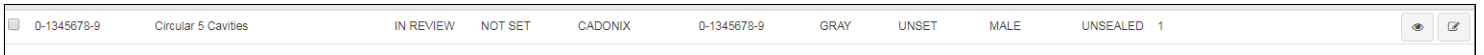


Fig 27.2. Quickstart Select button

Search the created connector **0-1345678-9 (Circular 5 Cavities)** within the Quickstart Library then click the **Edit**



command located at the right of the description.



Next we will add our Loading view image, Under **Connector, Image Details, Loading** click the **Select/Upload File**

A blue button with a white upload icon and the text "Select/Upload File".

button browse within the **File Explorer** to **quickstart/Connector/** and double click on **5-Cavity-circular-Connector.svg** to select it. (Fig 27.3)

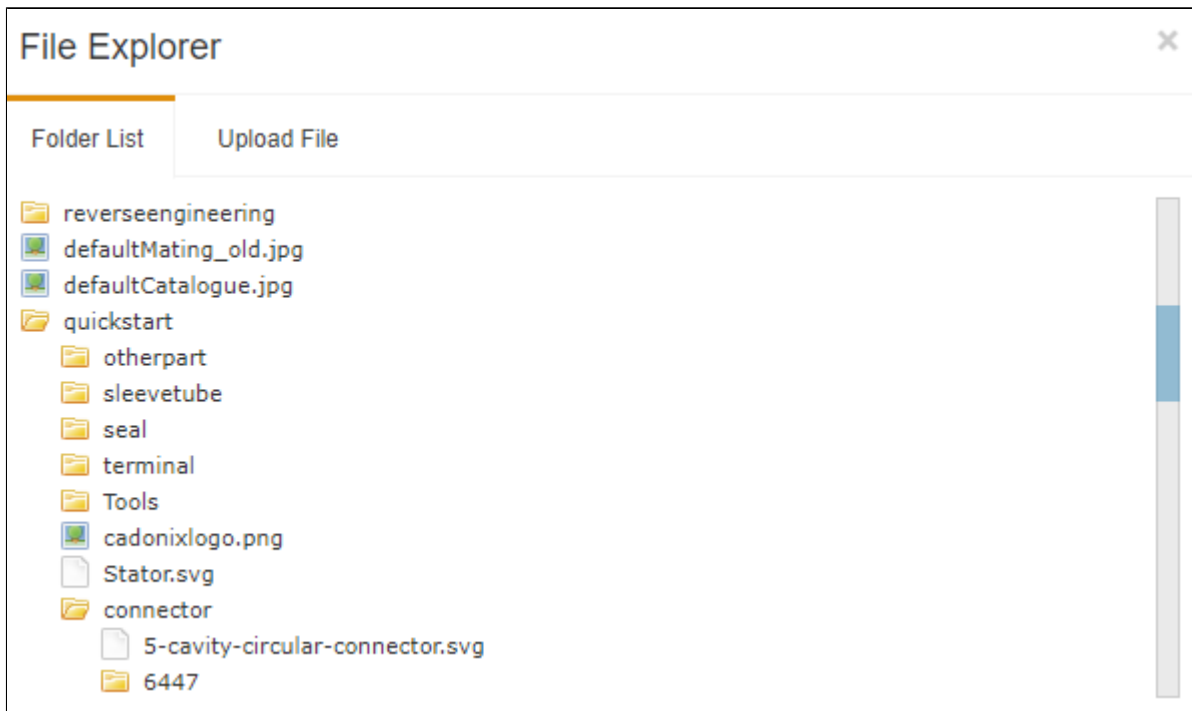


Fig 27.3 File Explorer, Local to Arcadia.

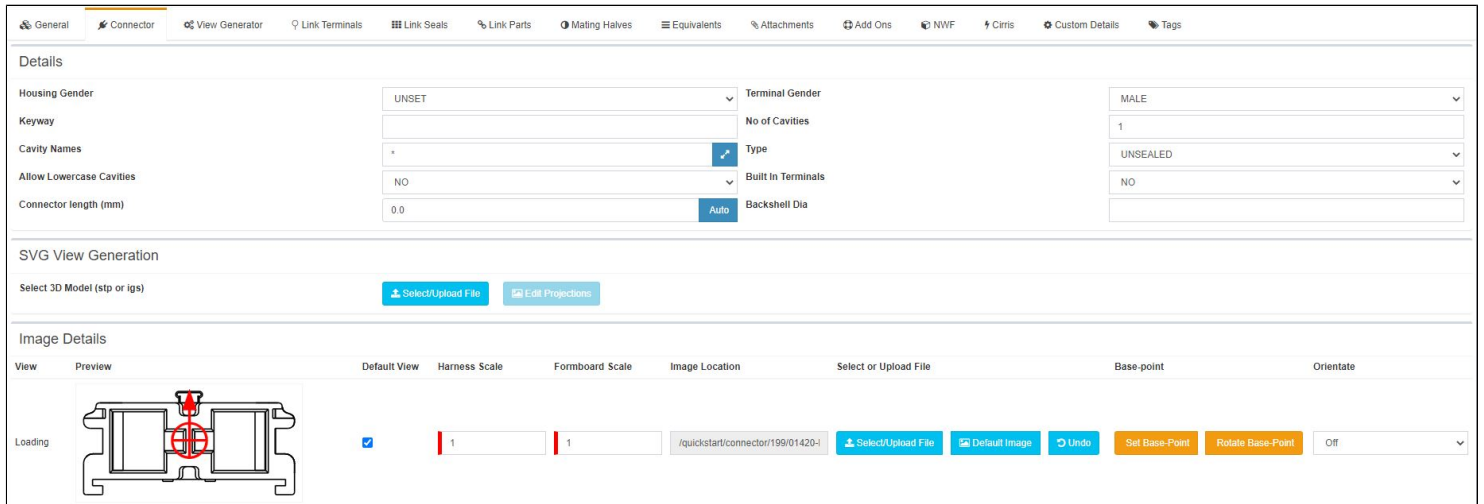


Fig 27.4 A connector Base-Point orientation can be set by clicking the "Rotate Base-Point".

You will now need to select "Set Base-Point"  , We can rotate the Base-Point based on the orientation of the connector.

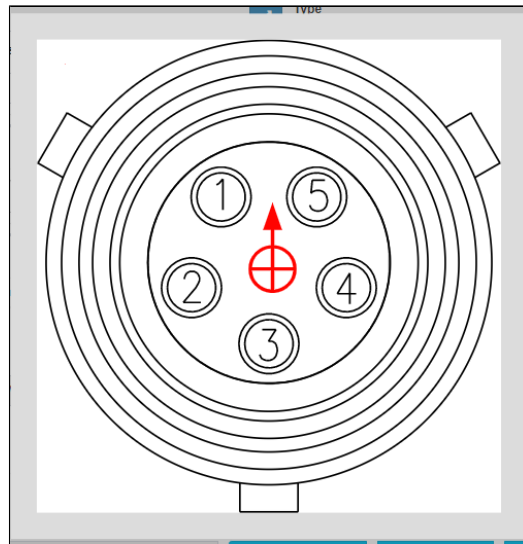


Fig 27.5 Image showing set Base-Point

Once completed this will now update the image thumbnail view so this is easy to view your selected base point.

**Fig 27.5**

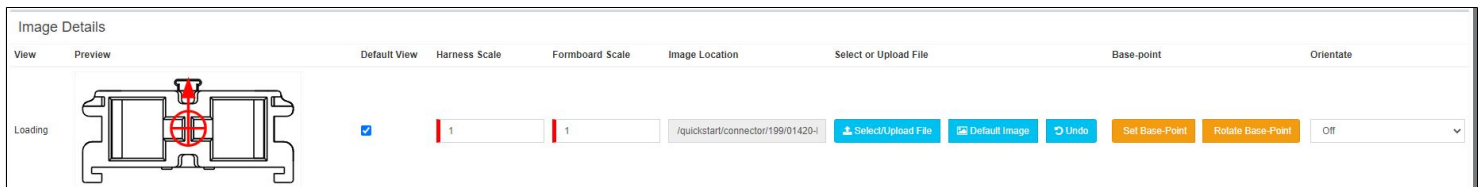


Fig 27.6 Thumbnail view of the image with Base-point.

Now when you place the component on to the Harness you will be able to place it accurately. (Note that all Quickstart connectors will have a base point already set)

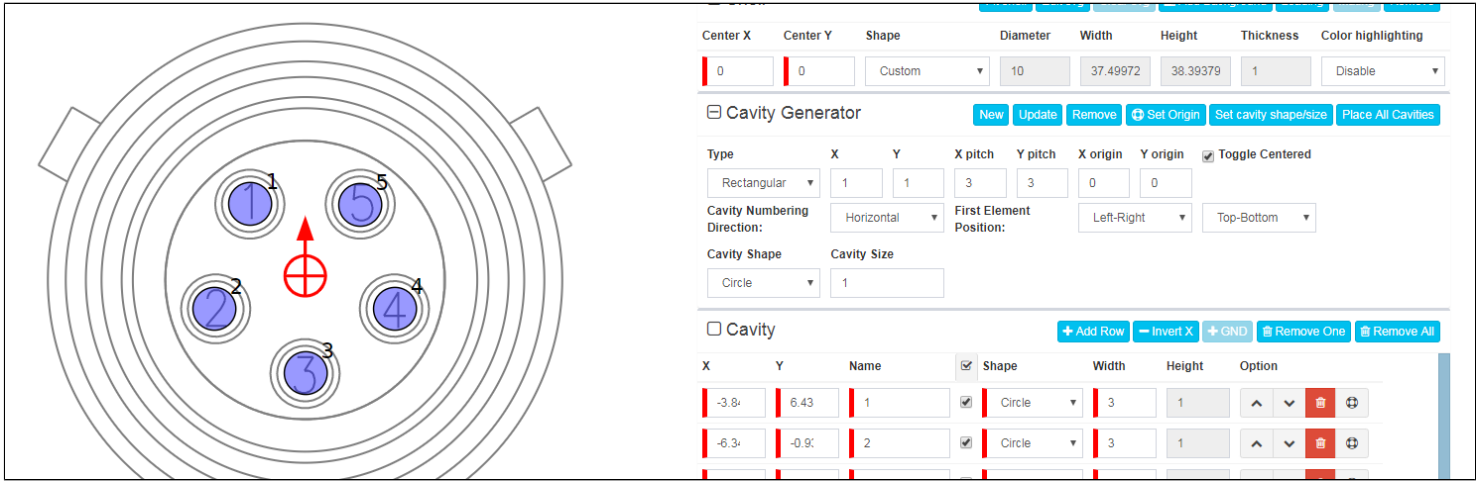
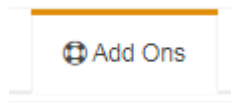


Fig 27.7 Image within the View Generator keeps the Base point to calculate the cavity lengths

Now we are going to automate the **Add-On** values for the wires into the connectors,

This will automatically calculate the distance between the base point and the individual cavity you have placed with the Highlighted Cavities,



To enable this please Edit the connector and click the **Add-On** tab along the connector information, Scroll down to the bottom of the Cavities and use the Calculate and update add-ons automatically

Calculate and update addons automatically

another prompt will show requiring the Length inside the connector, The default value for this is 10mm (For our example we will stick to 10mm)

This will now populate the cavity Add-on/Knock-off with a value.

Click update component and this will now be applied to the connector.

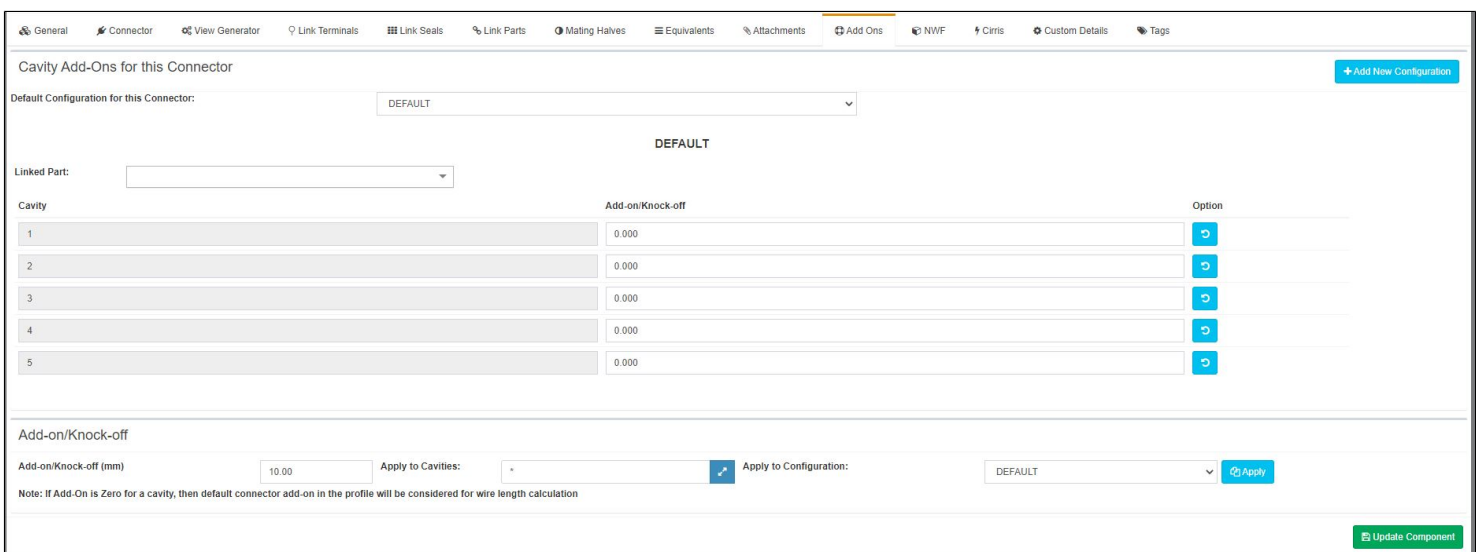


Fig 27.8 Add-on/Knock-off lengths are now calculated

## 28. Create a Generated View

The next step is to add generated views for our new Terminal and Connectors.

From the **Home** menu select **Component DB** (Fig 28.1) next select **Quickstart** (Fig 28.2)

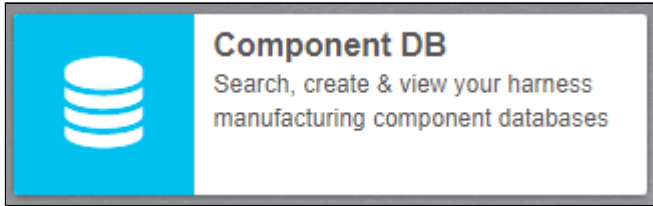


Fig 28.1. Component Database Select button

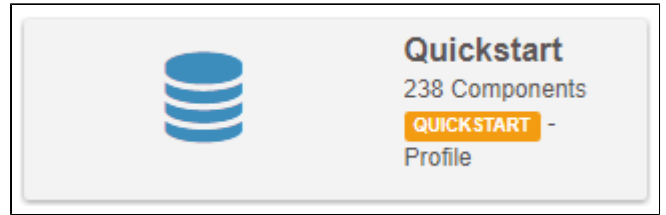





Fig 28.2. Quickstart Select button

Click the **Terminals**  Terminals (73) under Part number add 0-7654321-9 (The part number of our M6 Ring)

Click the **Edit**  button to open the components properties

Click the **Terminal**  Terminal tab and add the following details to to the terminal.


Terminal Gender: Male

Strip length (mm): 6.00

Wire CSA From (mm<sup>2</sup>): 3.00

Wire CSA To (mm<sup>2</sup>): 6.00

Next we will add our Top view image, Under **Image Details, Top** click the **Select/Upload File** button

 browse within the **File Explorer** to **quickstart/terminal/** and double click on **Ring-terminal.svg** to select it. (Fig 28.3)

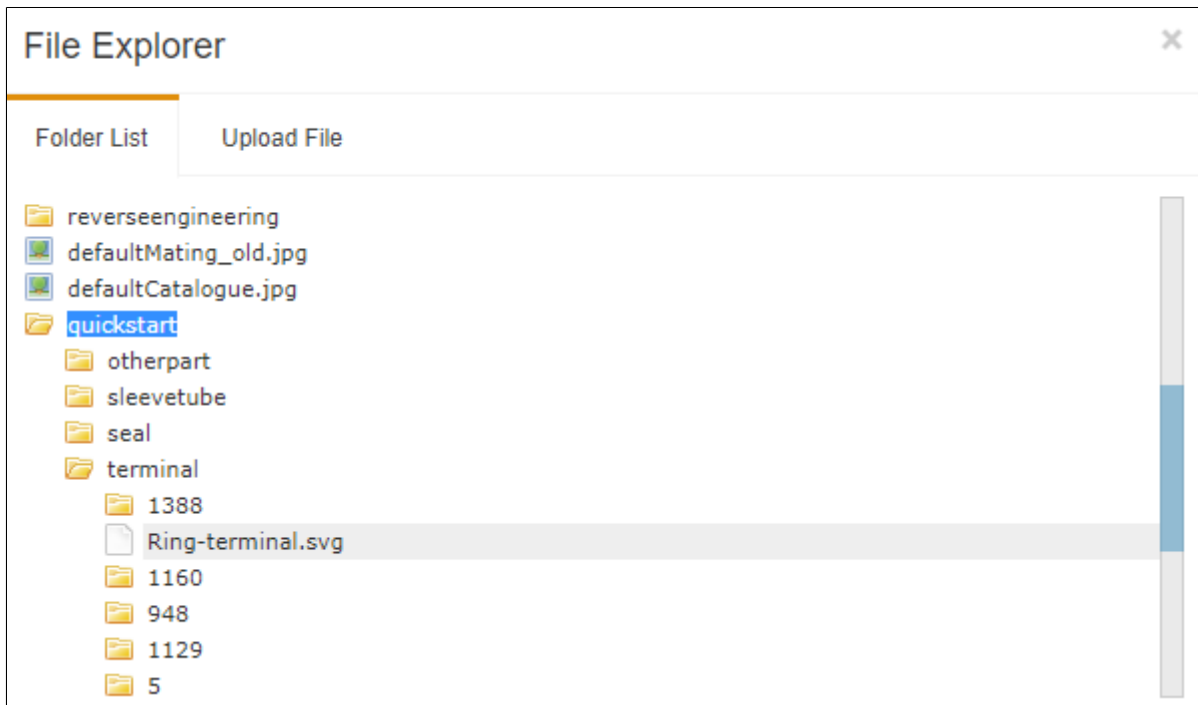



Fig 28.3 File Explorer file selection window

Scroll down till you can click **Update Component**  this saves the changes we have just made.

We will next create a generated view for our Square Connector.

From the **Home** menu select **Component DB (Fig 28.4)** next select **Quickstart (Fig 28.5)**

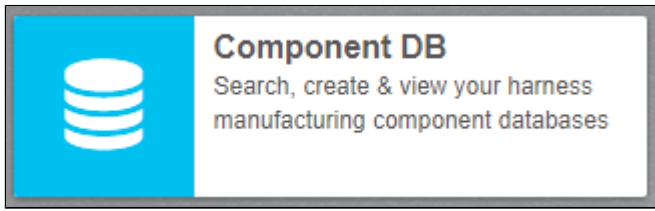


Fig 28.4. Component Database Select button

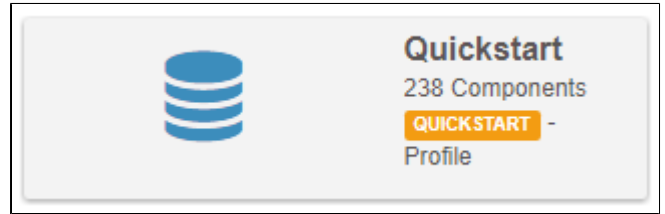


Fig 28.5. Quickstart Select button

Click the **Terminals** **Connectors (35)** under Part number add 0-1234567-9 (The part number of our Square 6 Cavity connector)

Click the **Edit** button to open the components properties

Click the **Connector** **Connector** tab and add the following details to to the terminal.

Housing Gender: UNSET

Terminal Gender: MALE

No of Cavities: 6

Next we will add our Top view image, Under **Image Details, Top** click the **Select/Upload File** button

browse within the **File Explorer** to **quickstart/Connector/** and double click on **6-Cavity-Connector.svg** to select it. (Fig 28.6)

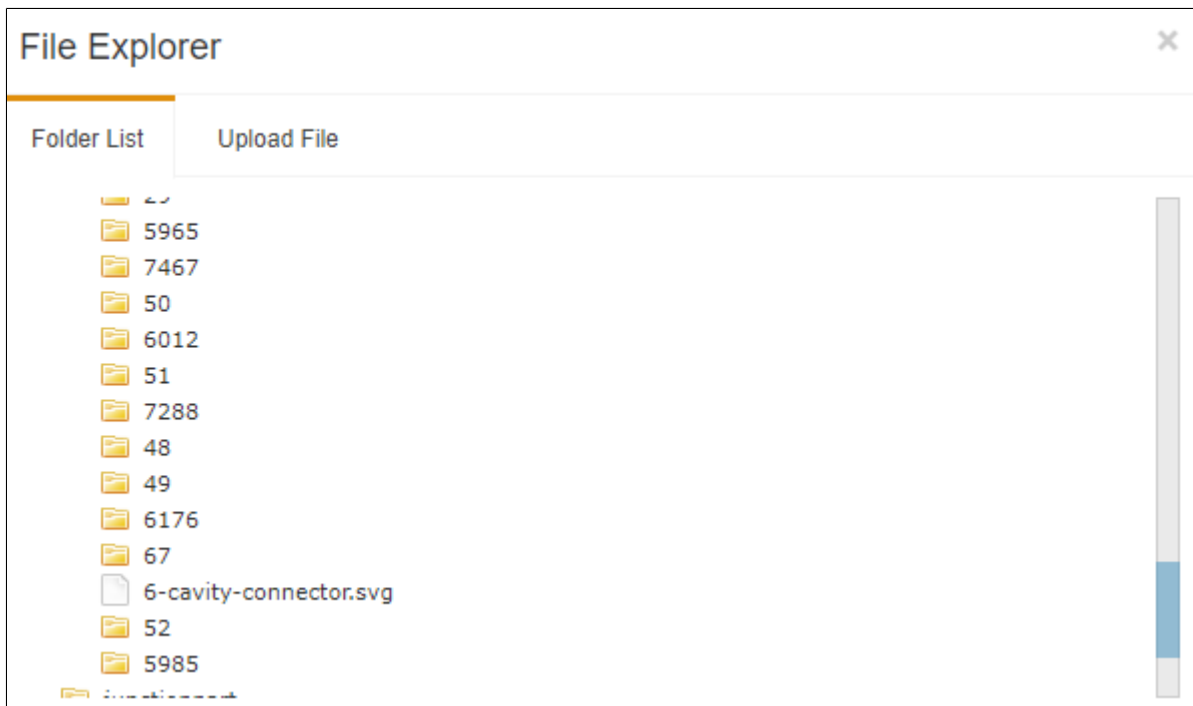


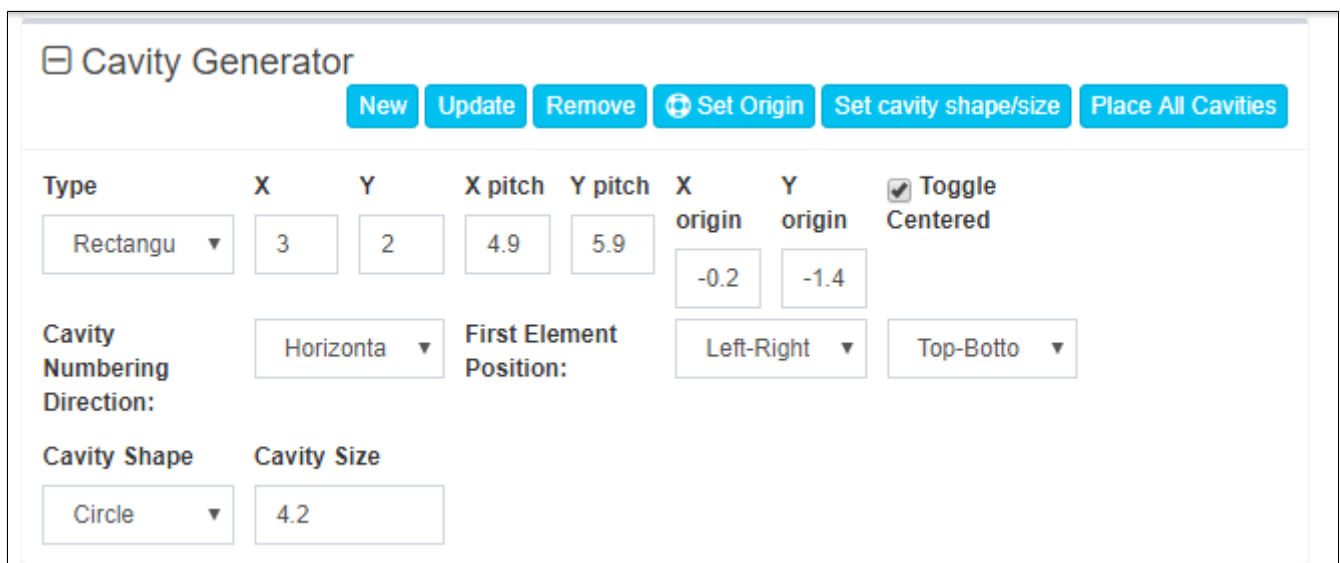
Fig 28.6 File Explorer file selection window

Click the **View Generator** **View Generator** tab under the **Shell** section click **Add Background** and browse within the **File Explorer** to **quickstart/Connector/** and double click on **6-Cavity-Connector.svg** to select it. (Fig 28.6) We next need to add our 6 cavities to the image, Under the **Cavity Generator** section set the following Values. (Fig 28.7) Type : Rectangular (Sets the pattern for the cavities)

X : 3 (Number of cavities in the horizontal row)  
Y : 2 (Number of cavities in the vertical row)  
X Pitch : 4.9 (Spacing on the horizontal axis)  
Y Pitch : 5.9 (Spacing on the vertical axis)  
Cavity Numbering Direction : Horizontal  
First Element Position : Left-Right, Top-Bottom  
Cavity Shape : Circle  
Cavity Size : 4.2

Click the **New** New button to add the cavities to the image.

Click **Place All Cavities** Place All Cavities to individually set the position of each of the 6 cavities, left click to confirm each location. **(Fig 25.8)**



The screenshot shows the 'Cavity Generator' interface with the following settings:

- Buttons:** New, Update, Remove, Set Origin, Set cavity shape/size, Place All Cavities
- Type:** Rectangu (dropdown)
- X:** 3
- Y:** 2
- X pitch:** 4.9
- Y pitch:** 5.9
- X origin:** -0.2
- Y origin:** -1.4
- Toggle Centered:**
- Cavity Numbering Direction:** Horizontal (dropdown)
- First Element Position:** Left-Right (dropdown), Top-Botto (dropdown)
- Cavity Shape:** Circle (dropdown)
- Cavity Size:** 4.2

Fig 28.7 Cavity Generator options

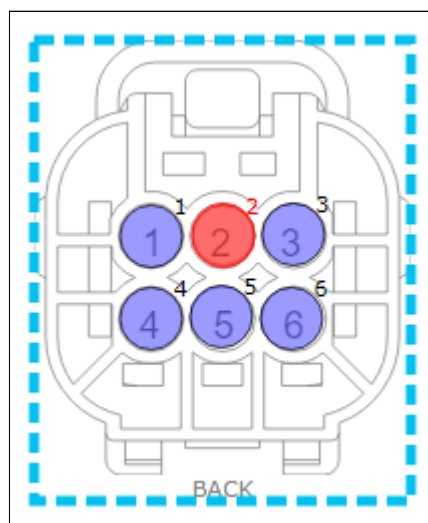


Fig 28.8 Cavity Placement

Scroll down till you can click **Update Component** Update Component this saves the changes we have just made.

We will next create a generated view for our Circle Connector.

From the Home menu select **Component DB** (Fig 28.9) next select **Quickstart** (Fig 28.10)

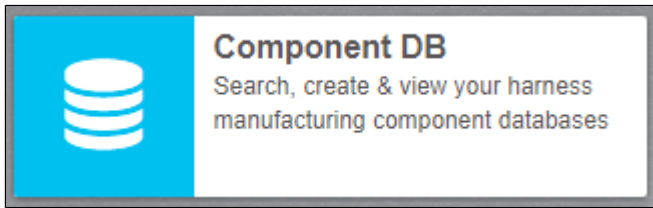


Fig 28.9. Component Database Select button

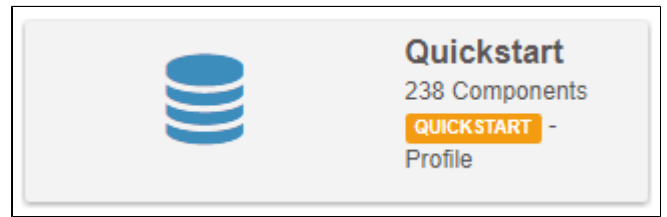


Fig 28.10. Quickstart Select button

Click the **Terminals** **Connectors (35)** under Part number add 0-1345678-9 (The part number of our Circle 5 Cavity connector)

Click the **Edit** button to open the components properties

Click the **Connector** **Connector** tab and add the following details to to the terminal.

Housing Gender: UNSET

Terminal Gender: MALE

No of Cavities: 6

Type: SEALED

Next we will add our Top view image, Under **Image Details, Top** click the **Select/Upload File**

button browse within the **File Explorer** to **quickstart/Connector/** and double click on **5-Cavity-circular-Connector.svg** to select it. (Fig 28.11)

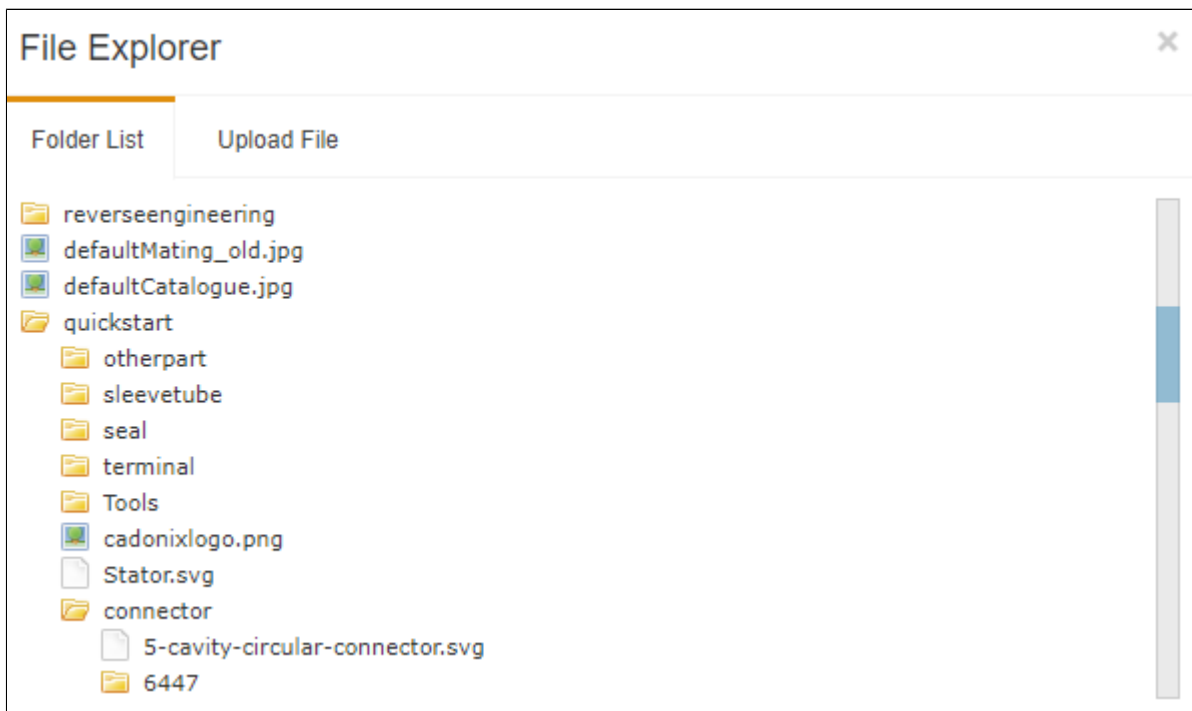


Fig 28.11 File Explorer file selection window

Click the **View Generator** tab **View Generator** under the **Shell** section click **Add Background** and browse within the **File Explorer** to **quickstart/Connector/** and double click on **6-Cavity-Connector.svg** to

select it. **(Fig 28.11)** We next need to add our 5 cavities to the image, Under the **Cavity Generator** section set the following Values. **(Fig 28.12)**

- Type : Circle (Sets the pattern for the cavities)
- X : 5 (Number of cavities in the circle)
- Radius : 4 (dimensions of the circle)
- Cavity Numbering Direction : Clockwise
- Cavity Shape : Circle
- Cavity Size : 19

Click the **New** New button to add the cavities to the image.

Click **Place All Cavities** Place All Cavities to individually set the position of each of the 5 cavities, left click to confirm each location. **(Fig 26.13)**

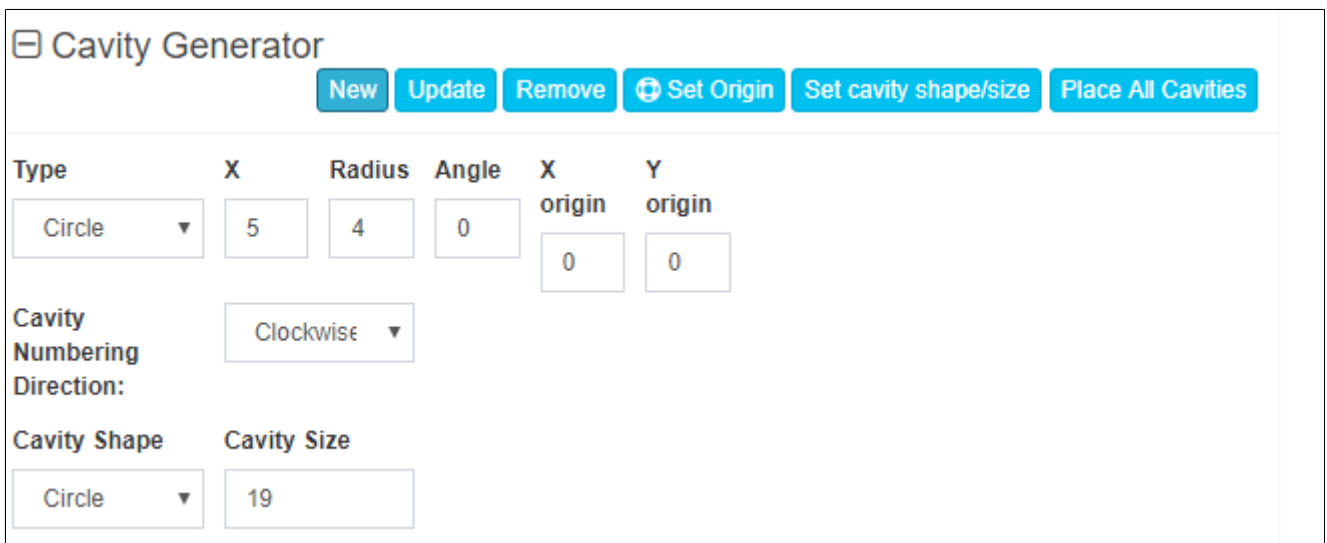


Fig 28.12 Cavity Generator options

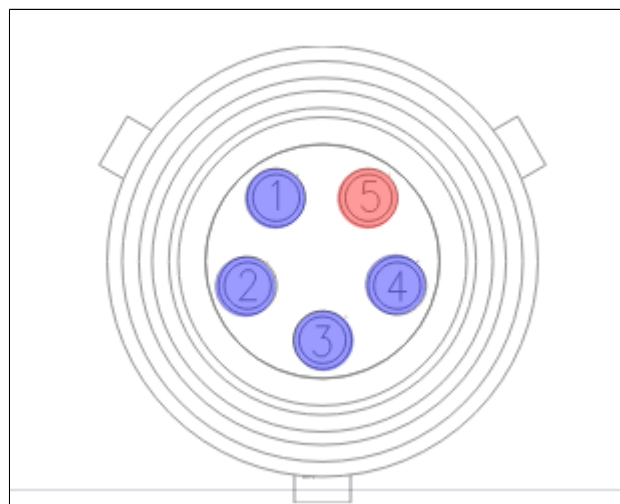


Fig 28.13 Cavity Placement

Scroll down till you can click **Update Component** Update Component this saves the changes we have just made.

## 29. Create a Splice Configuration & Splice

You will now set up a Splice configuration as well as creating a Splice and assigning the Splice Configuration to that splice.

From the **Home Menu (Fig 29.3)** select **Settings** then click **Company Profiles (Fig 29.4)** next select the **"Quickstart" Profile (Fig 29.5)**

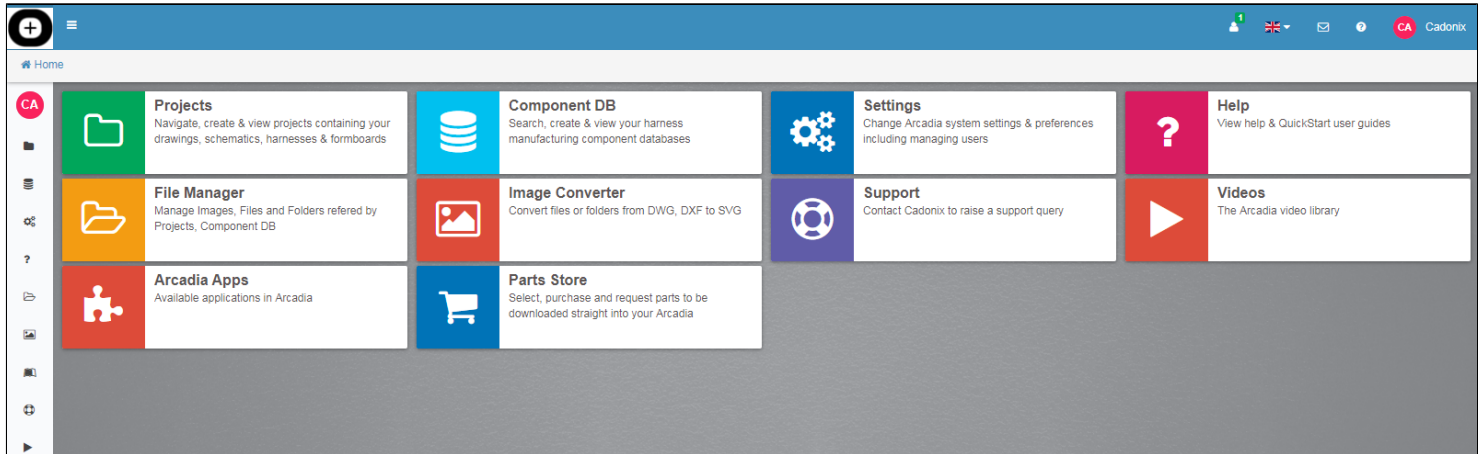


Fig 29.3 Home Menu

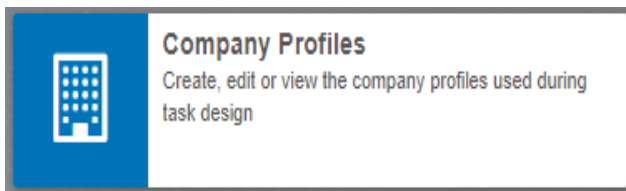


Fig 29.4 Click the Profile to Enter

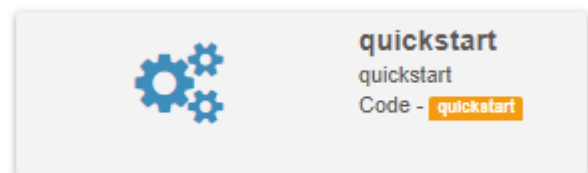


Fig 29.5 Click Quickstart

You can now create a Splice Configuration, found within the Quickstart Profile: **Quickstart > Component DB > Splice Configuration**.

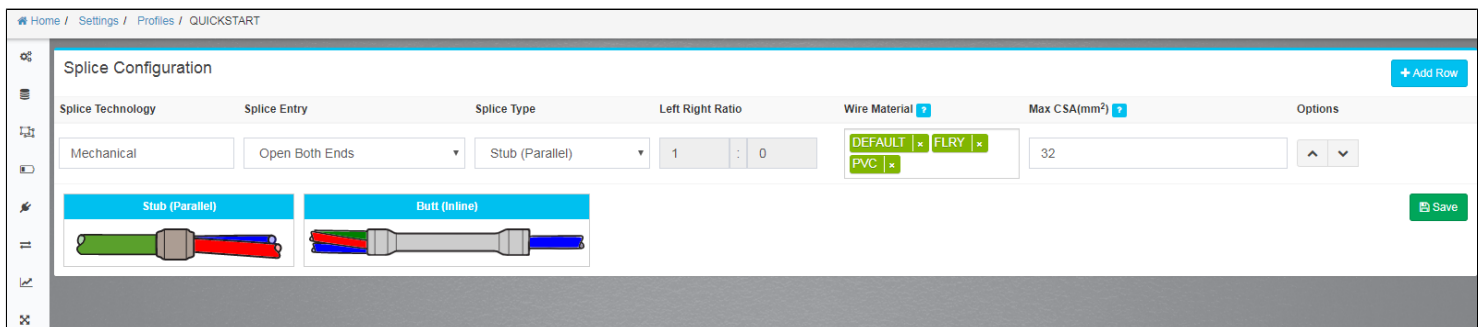


Fig 29.6: You can also input this information for each Splice manually, Splice Configuration will help Automate the Process.

Create a **Splice Configuration** as a **Mechanical** splice technology, **Open Both Ends** with a splice type set to **Stub &** a Maximum CSA of **32** (Please select the *Wire Material* these are created within the Profile > General > Material Editor/Gauge Editor)

Once you have created your splice configuration you can now go back into the library and apply that to a Splice.

Now you will create your **Splice** and assign your new **Splice Configuration**.

From the **Home** menu (Fig 29.7) select **Component DB (Fig 29.8)** next select the **"Quickstart" Profile (Fig 29.9)**

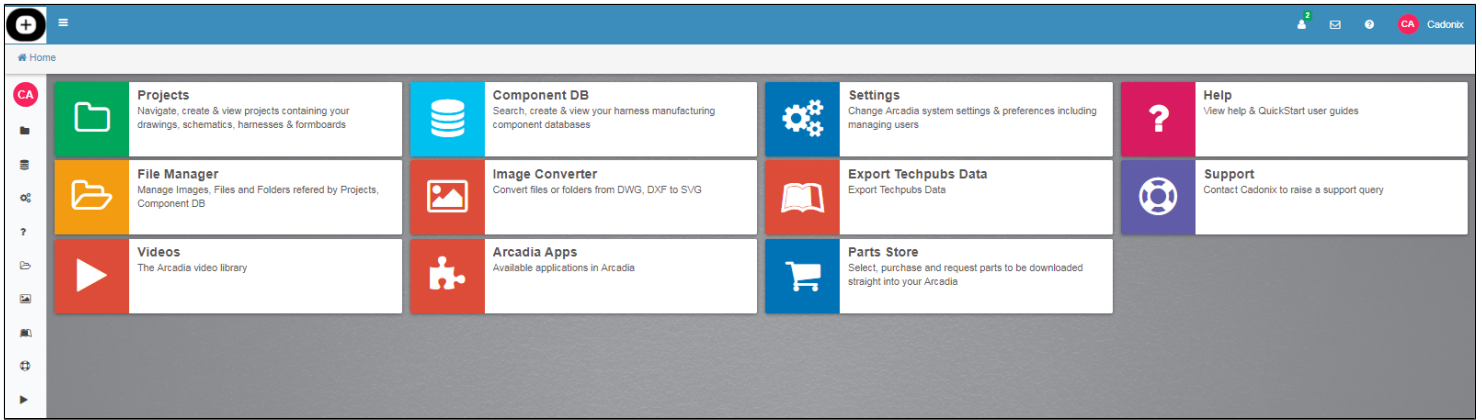


Fig.29.7 Home Menu



Fig 29.8. Component Database Select button.

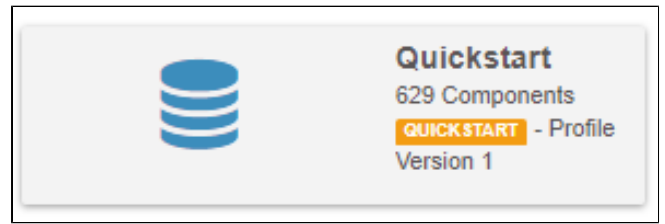
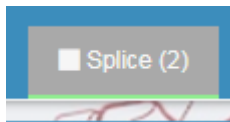


Fig 29.9. Quickstart Database Select button.



Click on the **Splice Tab**



and then select Add New Component



To add a new Splice Component.

Description: TERM BUTT 14-16  
 Part Number: 2140014  
 Company: CADONIX

Fig 29.10 General Connector Information.

Once we have completed our General information please click on the Splice tab to populate the Splice Technology, Select Mechanical & The greyed out boxes will Automatically populate with the information you have populated,

Change the **Sealing Type** to **Heatshrink**  
**Strip Length: 7.62**  
**Length for Heatshrink Covering (mm): 38.1**

General Splice Applicator Link Parts Equivalents Attachments Custom Details NWF Tags

Details

Splice Technology  Splice Type

Splice Sealing Type HEATSHRINK Finish TIN

Splice Dia (mm) 0 Splice Add-on/Knock-off (mm) 0

Strip Length (mm) 0 Length for Heat Shrink Covering (mm) 39.4

Splice Loopback Length (mm) 0 Left Right CSA Ratio

Splice Entry

Splice Configuration

Wire Material COPPER Splice Configuration

A Side CSA (mm<sup>2</sup>) 0.35 B Side CSA (mm<sup>2</sup>) 2

SVG View Generation

Select 3D Model (stp or igs)

Image Details

View Preview Default View Harness Scale Formboard Scale Image Location Select or Upload File Base-point Orientate

Top  1 1 /quickstart/splice/2844

Fig 29.11 Splice Information.

Now click  and head back to the Quickstart Schematic.

---

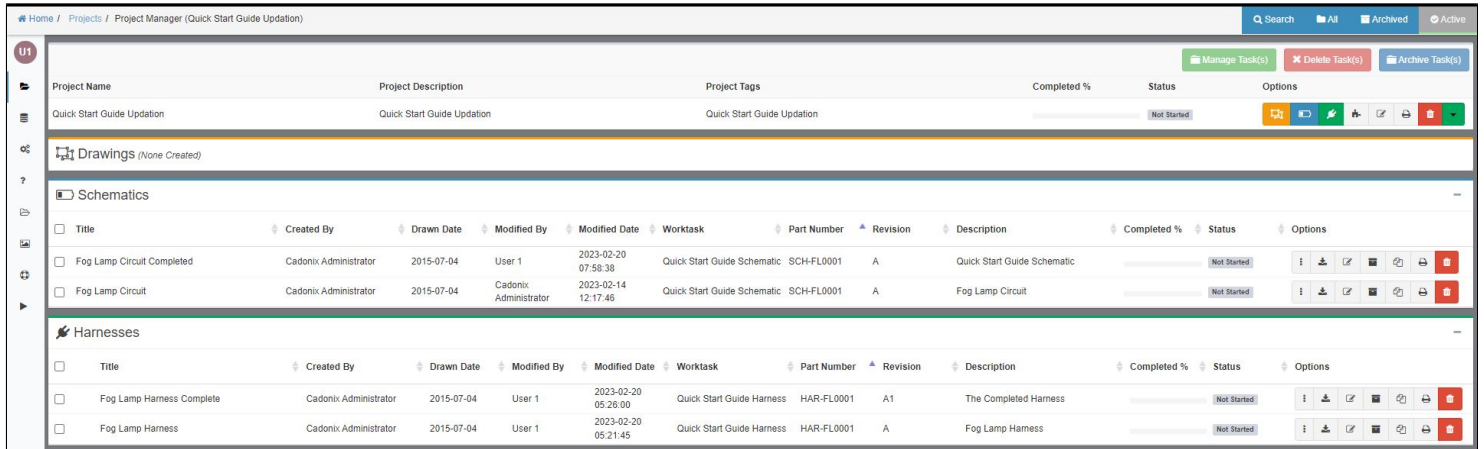
## Harness Layout Quick Start

You will now use the Harness commands in Arcadia to finish off the Fog Lamp Harness Layout.

➔ **Note:** Harness is an optional module and may or may not be included in your Arcadia edition.

### 30. Open a Harness Layout

➔ **Note:** If you have been working on the Fog Lamp circuit, return to the Project Details screen by clicking on the "Exit" toolbar and selecting "Exit Drawing".



The screenshot shows the Project Manager interface for a project named "Quick Start Guide Update". The interface is divided into several sections: "Drawings (None Created)", "Schematics", and "Harnesses". Each section contains a table of items with columns for Title, Created By, Drawn Date, Modified By, Modified Date, Worktask, Part Number, Revision, Description, Completed %, Status, and Options. The "Schematics" table lists "Fog Lamp Circuit Completed" and "Fog Lamp Circuit". The "Harnesses" table lists "Fog Lamp Harness Complete" and "Fog Lamp Harness".

Title	Created By	Drawn Date	Modified By	Modified Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Circuit Completed	Cadonix Administrator	2015-07-04	User 1	2023-02-20 07:58:38	Quick Start Guide Schematic	SCH-FL0001	A	Quick Start Guide Schematic		Not Started	[Options]
Fog Lamp Circuit	Cadonix Administrator	2015-07-04	Cadonix Administrator	2023-02-14 12:17:46	Quick Start Guide Schematic	SCH-FL0001	A	Fog Lamp Circuit		Not Started	[Options]

Title	Created By	Drawn Date	Modified By	Modified Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Harness Complete	Cadonix Administrator	2015-07-04	User 1	2023-02-20 05:20:00	Quick Start Guide Harness	HAR-FL0001	A1	The Completed Harness		Not Started	[Options]
Fog Lamp Harness	Cadonix Administrator	2015-07-04	User 1	2023-02-20 05:21:45	Quick Start Guide Harness	HAR-FL0001	A	Fog Lamp Harness		Not Started	[Options]

*30.1 Choose the drawing you which to work with, note this is spit down by module. Drawing, Schematic and Harness.*

Click on the title **Fog Lamp Harness**. You have now opened the Fog Lamp Harness.

## 31. The Harness Window

Take a moment to look at the different areas of the Schematic Window (**Fig 31.1**).

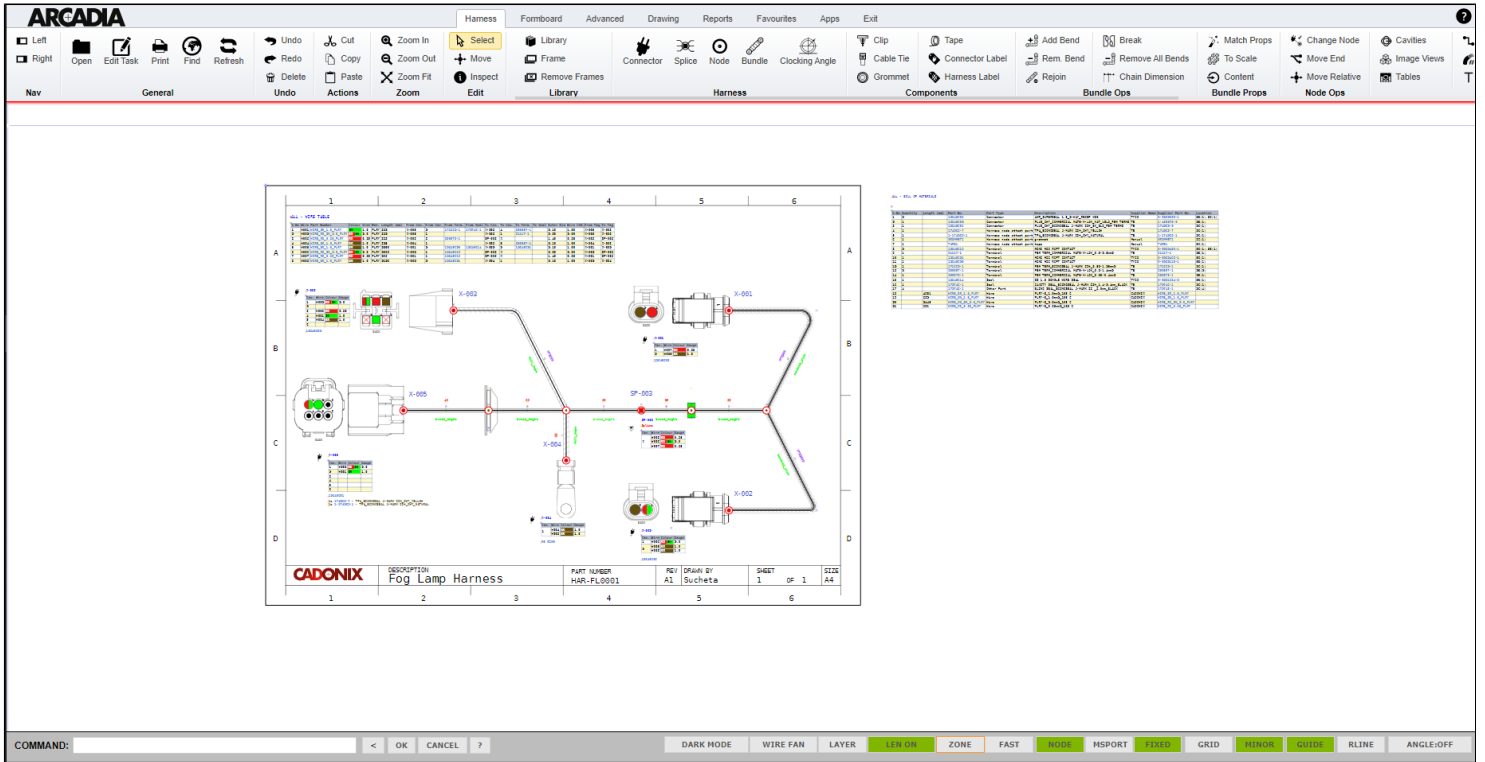
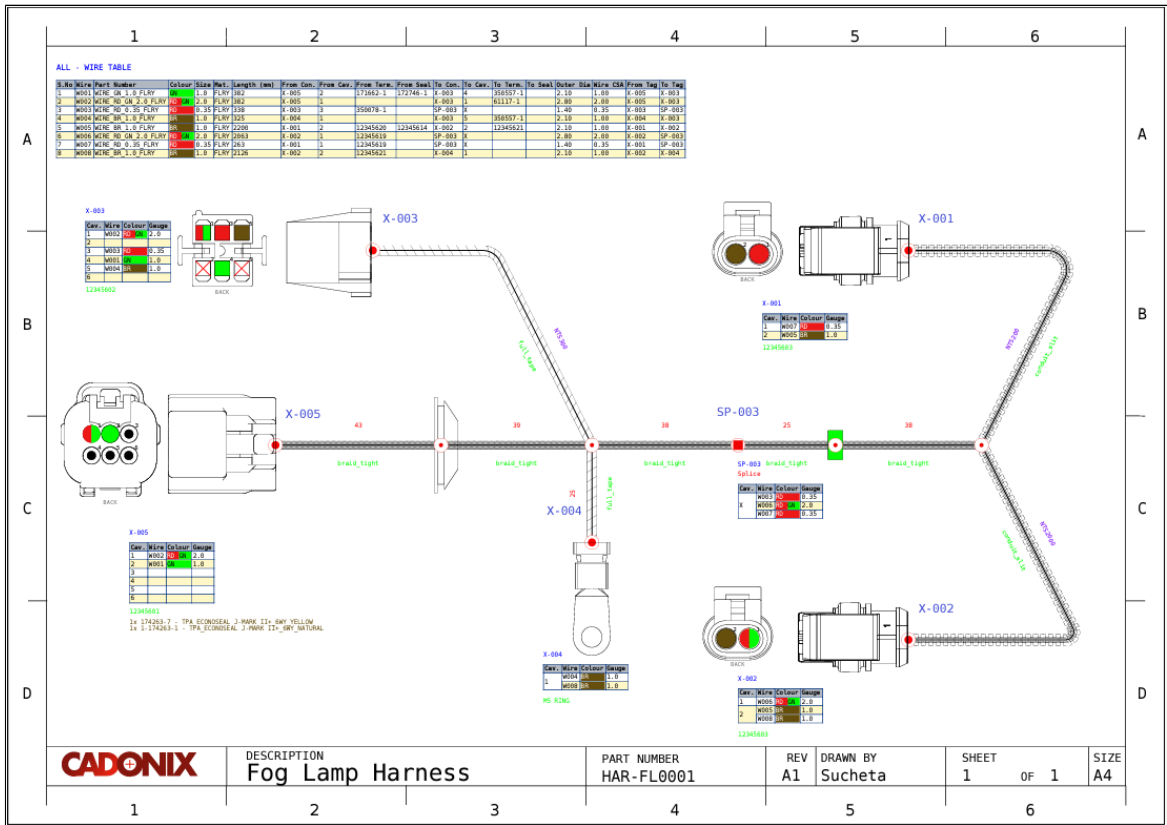


Fig 31.1 The Harness Window

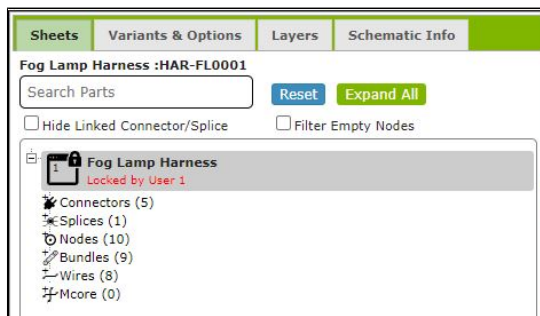
### 1 Toolbar



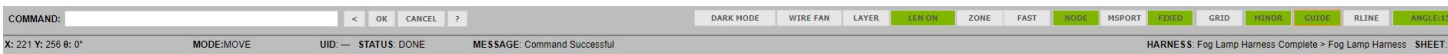
### 2 Worksheet



### 3 Project Navigator



### 4 Command, Message & Construction Toolbar



## 32. Mouse & Keyboard Commands



<b>D</b>	Dimension Shortcut
<b>F</b>	Insert Frame Shortcut
<b>J</b>	Arc Shortcut
<b>K</b>	Insert Leader
<b>L</b>	Line Shortcut
<b>M</b>	Move
<b>O</b>	Open Library
<b>Q</b>	Hide Project Navigator
<b>R</b>	Rectangle Shortcut
<b>T</b>	Text Shortcut
<b>U</b>	Curve Shortcut
<b>V</b>	Add Vertex to Polyline/Polygon
<b>W</b>	Hide Component Text Toggle
<b>X</b>	Pan
<b>Y</b>	Polygon Shortcut
<b>Z</b>	Zoom
<b>Del</b>	Delete Object
<b>Esc</b>	Quit current command

Advanced Keyboard shortcut details are available for additional functions by clicking the "?" link next to the command line at the bottom of the window.

COMMAND:

<

OK

CANCEL

?



**Left Button**

Used to select items

**Right Button**

Used to open the context menu

**Centre Wheel / Button**

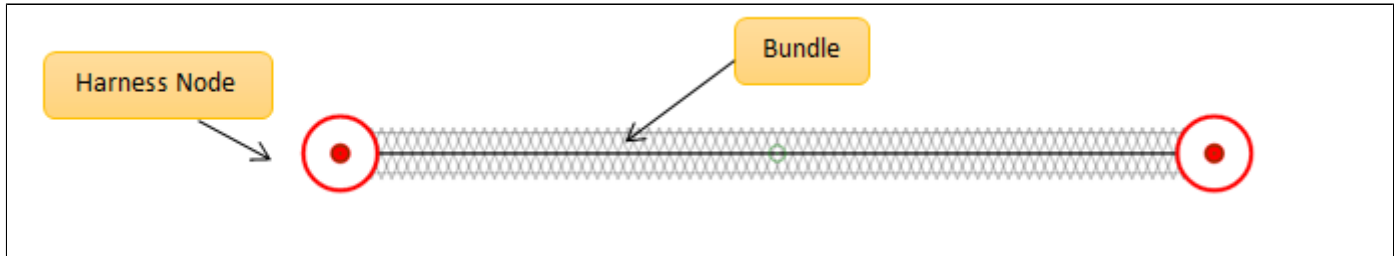
Used to Zoom when scrolled, Pan when pressed



### 33. Insert a Bundle

Bundles are used in Arcadia to represent physical groups of wires which run together. A bundle can contain one or many wires. Coverings such as tape, tubing and braid can be applied to a bundle and can also be layered.

Bundles are created between **Harness Nodes**.



*Fig 33.1 Harness Bundle*

You will now add the following bundle to the Fog Lamp Harness.

Click on the **Create Bundle** command

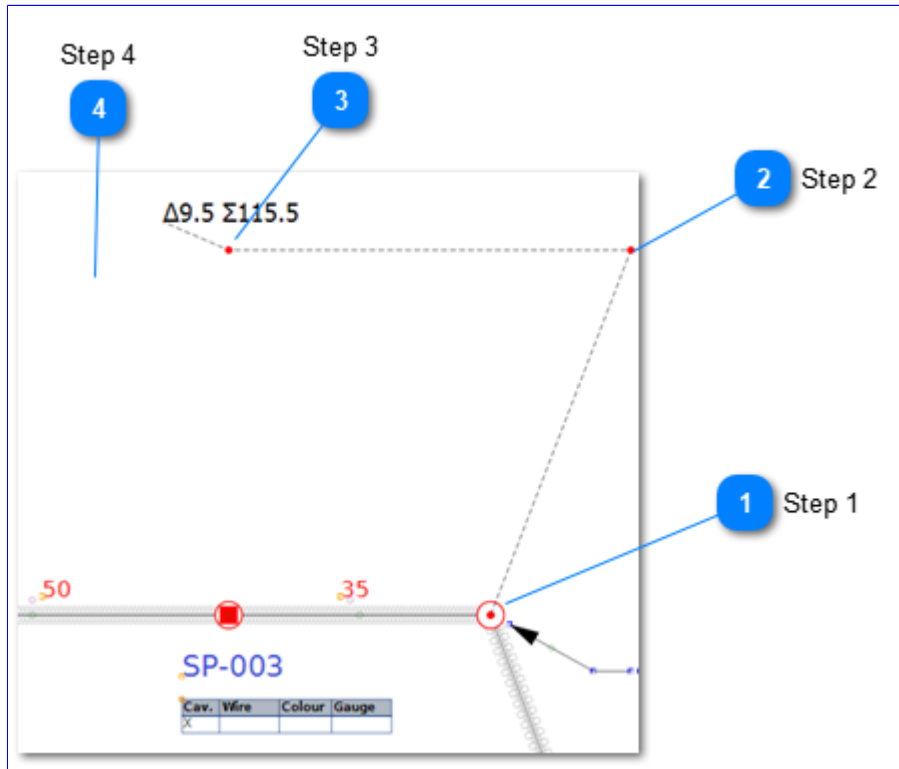


Fig 33.2 Placing a Bundle

#### 1 Step 1

Begin by **Left** clicking on the **Harness Node** at step 1. As you draw your bundle a dotted line will appear where the bundle will be drawn.

#### 2 Step 2

Next, **Left** click again at step 2.

#### 3 Step 3

Now drag out your mouse and **Left** click again at step 3.

#### 4 Step 4

Complete your bundle by **Right** clicking and then choosing **Finish**. Use **Esc** to quit the bundle command.

Note\* Each left click is setting a distance from the previous click.

➔ Tip: When right-clicking you can use predefined lengths to quickly enter a length of bundle or provide a custom length value.

---

Add a radius to the bend by clicking **Inspect**  then clicking on the **Bundle Control Node**.




*Fig 33.3 Bundle Control Node*

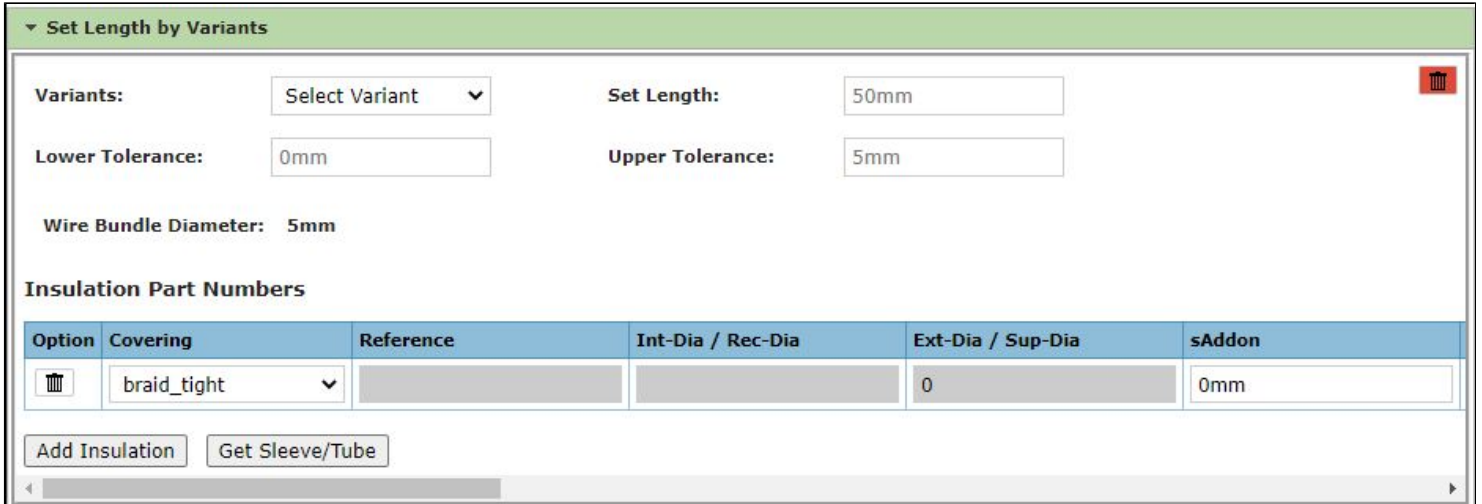
The **Bundle Properties** pane appears. Set the **Bend Radius** to **5**. Press **Submit** at the bottom of the screen.

➔ Tip: Control Nodes can be turned off or on by pressing **SHIFT + N**

### 34. Inspect Bundle & Add Insulation

You will now inspect the Bundle and add a covering, To add covering there are two different methods

1. Click the  icon then click on the **Bundle**,



The screenshot shows a software interface for configuring bundle insulation. It includes a 'Set Length by Variants' header, input fields for 'Set Length' (50mm), 'Lower Tolerance' (0mm), and 'Upper Tolerance' (5mm). A 'Wire Bundle Diameter' is set to 5mm. Below is a table for 'Insulation Part Numbers' with columns for Option, Covering, Reference, Int-Dia / Rec-Dia, Ext-Dia / Sup-Dia, and sAddon. The 'Covering' dropdown is set to 'braid\_tight' and 'sAddon' is 0mm. Buttons for 'Add Insulation' and 'Get Sleeve/Tube' are at the bottom.

Option	Covering	Reference	Int-Dia / Rec-Dia	Ext-Dia / Sup-Dia	sAddon
	braid_tight			0	0mm

Fig 34.1 Inspect Bundle

The **Bundle Properties** appears. Under the section **Insulation Part Numbers**, use the **Covering** drop-down list and select **conduit\_slit** (Fig 34.1).

2. You can Right Click the bundle to quick add Covering from your Library.

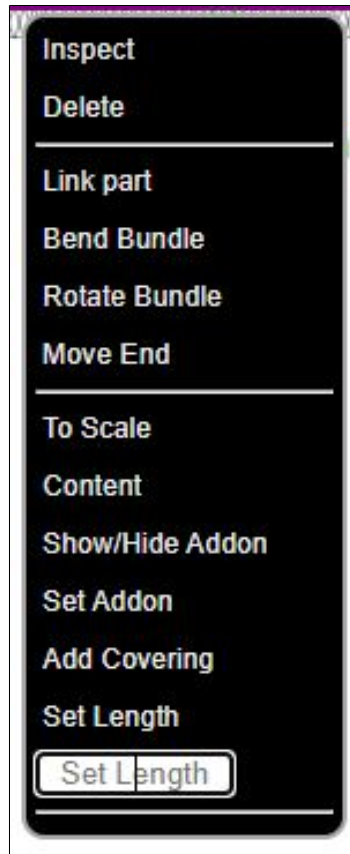


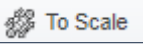
Fig 34.2 Right click menu for Bundle

➔ **Note:** It is possible to add a part number for the covering from the **Bundle Properties** Pane.

---

On the Bundle Properties or when using the right click function, you will also set the bundle **Set Length** as **200**.

Click **Submit Or Hit Enter (If adding length from right click menu)**. The drawing updates with the **Not To Scale (NTS)** length added along the bundle.

➔ **Note:** You can change a Not To Scale (NTS) bundle to become To Scale. Click the **Fix Bundle Length**  **To Scale** command. Then click on a NTS bundle. Next, click on the Harness Node at the end of the bundle in the direction you wish the bundle to adjust. The bundle will now adjust to become To Scale.

## 35. Move Bundle/Harness

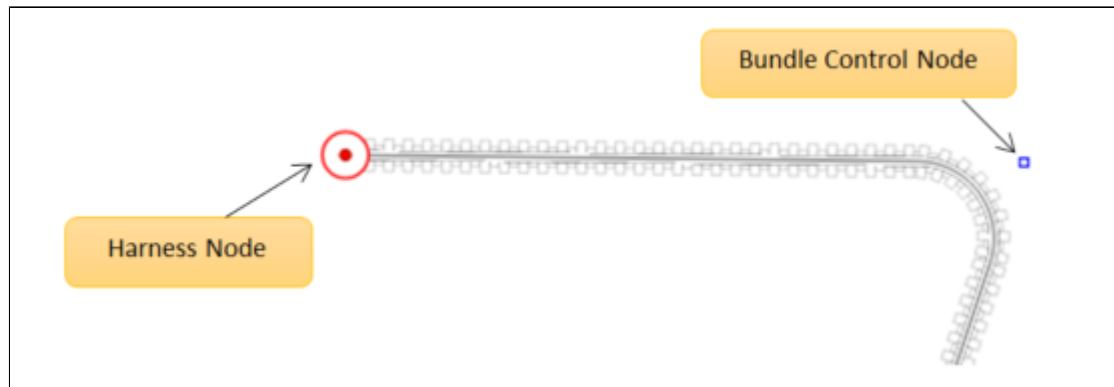







Fig 35.1 Move Bundle


You can move the bundle by using the **Move**  **Move** command. Select the Move command and click on either the Harness Node or the Bundle Control Node. This will allow you to pick up the bundle and move it accordingly.

You can move the Bundle End  **Move End** to any new Node destination if you wish to change the layout of your Harness during your design. Click the Bundle and then select the New node destination for the bundle.

Alternatively if you wish to change or move the Node you can do so with the Change Node  **Change Node** command. Click the Node you wish to move and then Click the new destination Node to apply the change.

Also we can move the Harness Relative  **Move Relative**. Simply click the Node you wish to move the harness relative to and place the Harness in its new location.

→ **Tip:** Use the Add Bend  **Add Bend** command to add additional bend to the bundle. Simply click on the Add Bend command then click on a section of the bundle to add a **Bundle Control Node** which can be used to drag out the bundle.

→ **Tip:** Use the Remove Bend  **Rem. Bend** command to remove a bend from a bundle. Simply click on the Remove Bend command and click on a **Bundle Control Node** for a bend.

## 36. Insert a Connector

You will now add a connector to the Fog Lamp Harness.

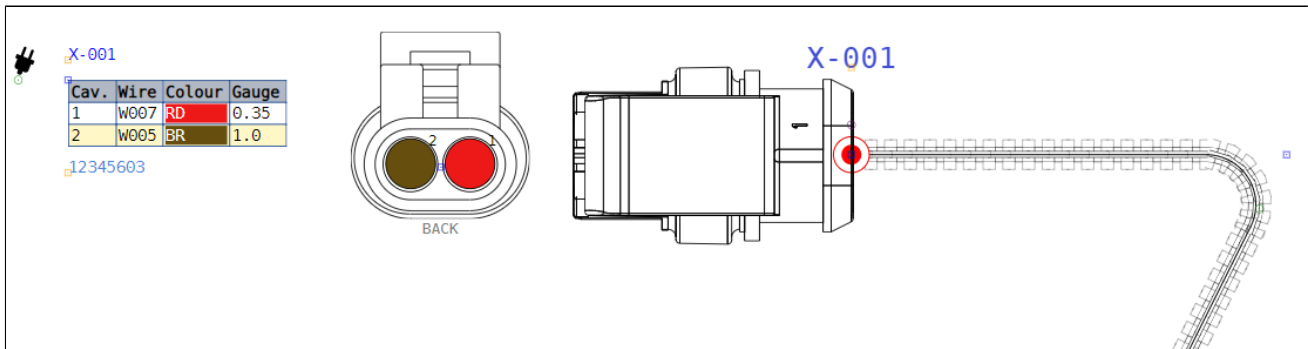



Fig 36.1 Insert Connector



Click on the Connector command on the toolbar  then click on the **Harness Node** at the end of the bundle. The **Create Connector Dialog** opens. You can select from a list of connectors which are stored in Component Libraries.

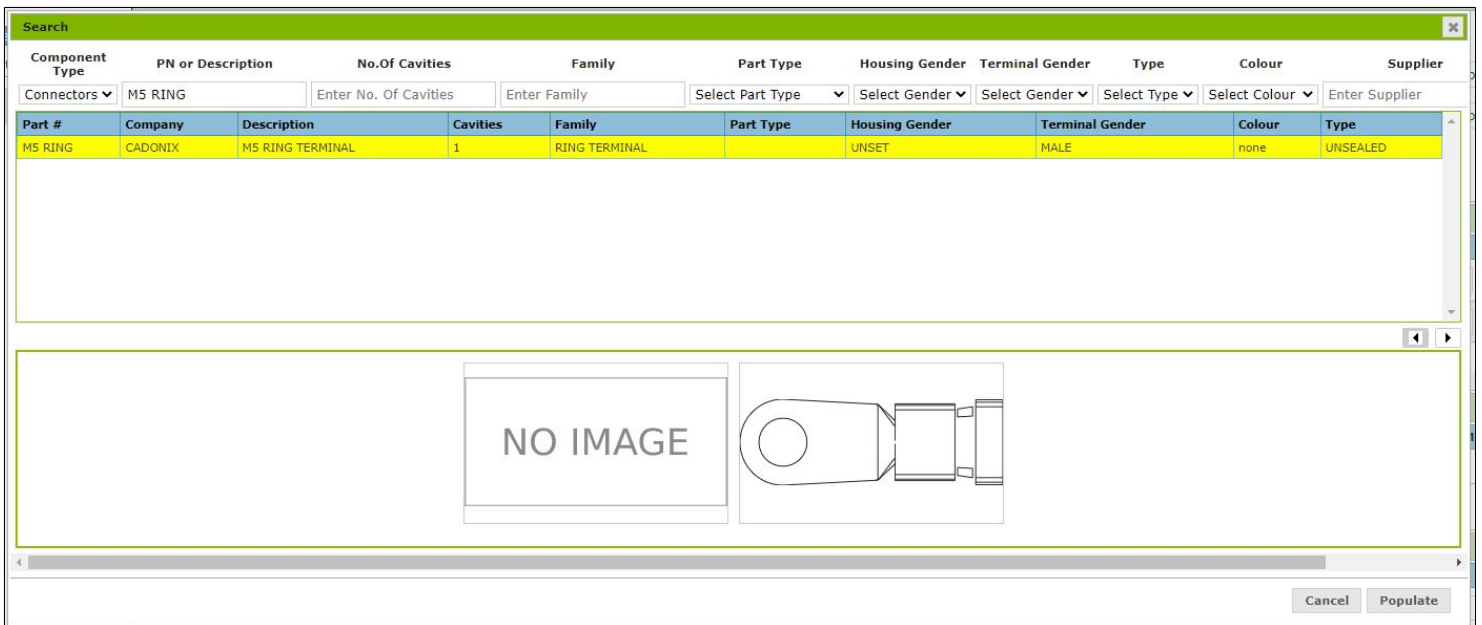



Fig 36.2 Create a Connector Dialog (Search Parts database)

➔ **Tip:** Acadia is supplied with a sample Component Library. You can create your own Component Libraries containing your own components.

Choose the **quickstart** library from the drop down menu. Click in the **Part Number** field.

Click the  icon to list all connectors in the library.

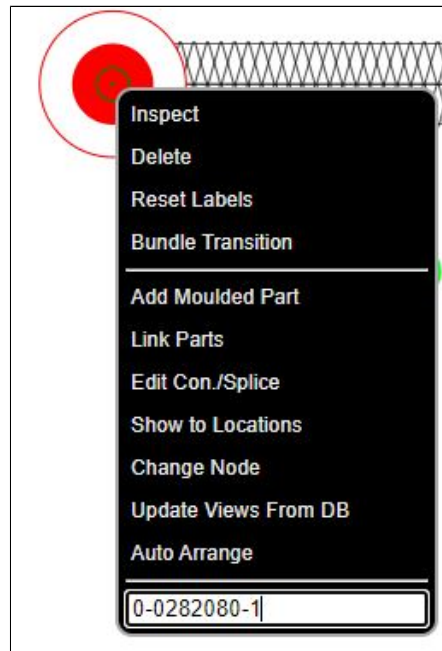
Select the **2 Cavity Super Seal Connector Part 0-0282080-1** in the list. Click **Populate**.

Click **Submit**. Place the Connector Table on the worksheet. The Connector symbol will automatically place it self in relation to the Connector not obstructing the Bundle, You can then choose to **Move** the connector if necessary.

The **Connector Properties** screen opens.

Set the Part Name field to read **X-001**. Click **Submit**.

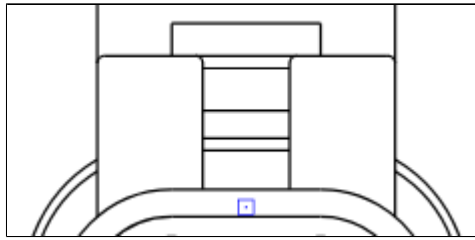
To quick add connectors Right click the node and enter the required Part number, This will automatically search the chosen database for the component, Press **Enter** to submit.



---

### 37. Inspect Component

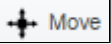
You can bring up the Component Properties using the **Inspect**  command and clicking on a Control Node. Control Nodes are the small blue squares.



*Fig 37.1 Connector control node*

---

## 38. Move a Component

Use the **Move**  command to move objects on the worksheet. To move an object, click on its **Control Node**.

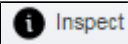
**Note:** You only need to click once on the control node to start the move not a click/hold drag.

➔ **Tip:** You can use the Move command to move Connector Tables and Connector Symbols. Ensure you click on their Control Nodes to move them.

➔ **Tip:** You can Hold CTRL before selecting the image to move the image by the center point of the image.

### 39. Insert a Wire Manually

Wires can be inserted manually or imported from a schematic in Arcadia if you have one. You will first add a wire manually to the Fog Lamp Harness.

Begin by clicking on the **Inspect**  command. Next, click on the **Connector Table Control Node** for connector **X-005**.

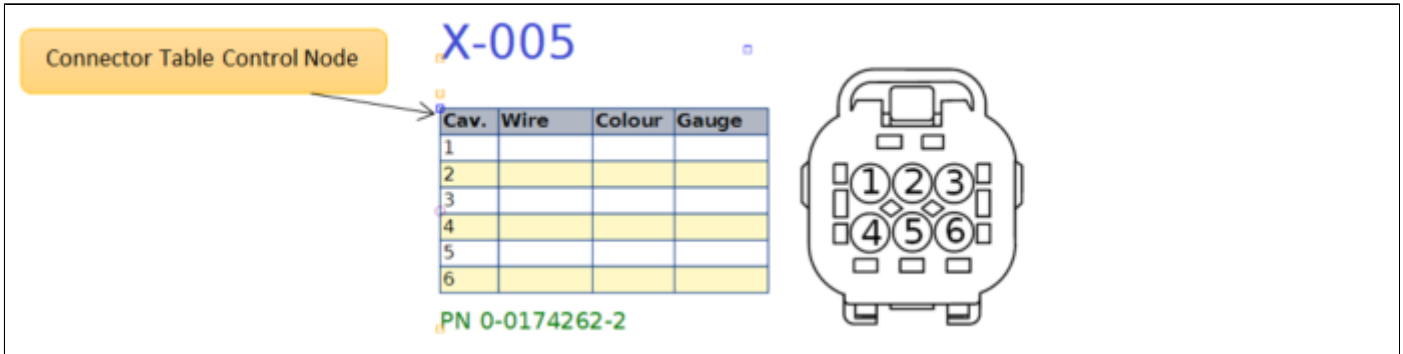



Fig 39.1 Insert Wire Manually Using the Connector Table

The **Connector Node Properties Pane** opens.

You will now add a wire to the Fog Lamp Harness. Under the section **Cavity Table** click the  icon next to **Cavity ID 1**.

Under the section **Wire Table**, you will now be able to add a wire from Cavity 1.

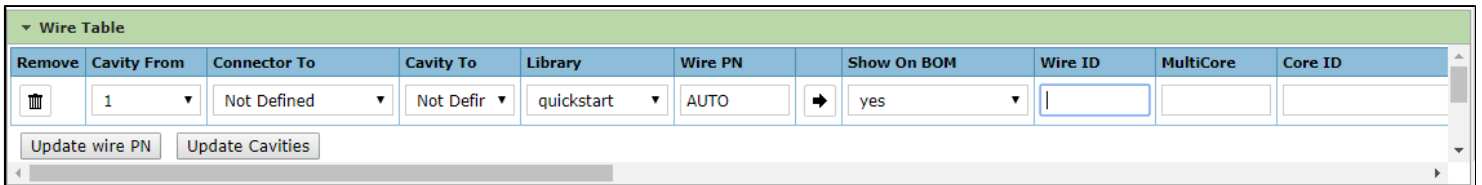



Fig 39.2 Wire Table

Enter the following values into the fields:

<b>Connector To</b>	SP-003
<b>Cavity To</b>	X
<b>Library</b>	quickstart
<b>Wire PN</b>	1.0_FLRY
<b>Primary Colour</b>	Red (RD)

Click the Search  Icon to find the wire in the library.

➔ Arcadia automatically finds the correct terminal and sealing part numbers for this connector.

Click **Submit**.

The Connector Table is updated with the wire. Navigate to the splice SP-003 and notice the result.



Also available in Arcadia is the **Wire Editor** a quick and easy way to view, edit and enter wires in a spreadsheet format.



From the Harness Screen Select Advanced Toolbar and click the **Wire Editor** tool to open the wire editor.

Wire Editor - Quick Start Guide Updation - Fog Lamp Harness Complete

Note: The changes to PreTerminated cable can be done only inside the harness drawing. The data remains readonly in the Editor.

	Wire ID	Signal Name	From Con	From Cav	From Con Desc	From Cav Display	From Func	To Con	To Cav	To Con Desc	To Cav Display	To Func	Material	Gauge	Primary Color	Secondary Color	Tertiary Color	Component DB	Wire C
1	W001		X-005	2				X-003	4				FLRY	1.0	GN	none	none	quickstart	Wire
2	W002		X-005	1				X-003	1				FLRY	2.0	RD	GN	none	quickstart	Wire
3	W003		X-003	3				SP-003	X	Splice			FLRY	0.35	RD	none	none	quickstart	Wire
4	W004		X-004	1				X-003	5				FLRY	1.0	BR	none	none	quickstart	Wire
5	W005		X-001	2				X-002	2				FLRY	1.0	BR	none	none	quickstart	Wire
6	W006		X-002	1				SP-003	X	Splice			FLRY	2.0	RD	GN	none	quickstart	Wire
7	W007		X-001	1				SP-003	X	Splice			FLRY	0.35	RD	none	none	quickstart	Wire
8	W008		X-002	2				X-004	1				FLRY	1.0	BR	none	none	quickstart	Wire
9																			
10																			

Fig 39.3 Wire Editor


The wire editor is a quick and easy way for the user to view, edit and export wires in the harness in a spreadsheet format.

The user can select existing Connectors and Cavities used in the Harness to enter the start and end destinations for each wire.

**Note:** typing AUTO for the Wire ID will automatically name the wire with the next available wire name when the user clicks **Save Wires**.

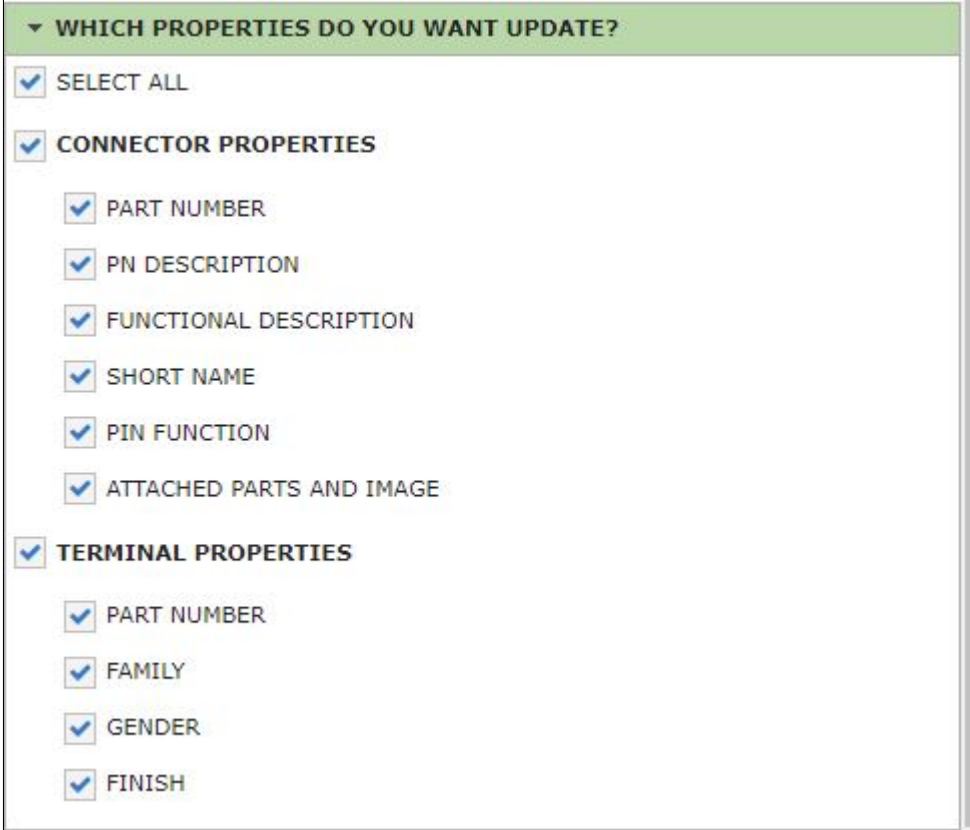
## 40. Load Wires from a Schematic

You will now use Arcadia to automatically load the wires from the Fog Lamp Schematic.

Click on the **Load Wires**  command. The **Load Wires Pane** appears.

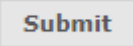
From the drop-down menus set **Project** to **Quick Start Project**, **Schematic** to **Fog Lamp Circuit**.

**Select All** when requested which properties do you want to update.



▼ WHICH PROPERTIES DO YOU WANT UPDATE?

- SELECT ALL
- CONNECTOR PROPERTIES**
  - PART NUMBER
  - PN DESCRIPTION
  - FUNCTIONAL DESCRIPTION
  - SHORT NAME
  - PIN FUNCTION
  - ATTACHED PARTS AND IMAGE
- TERMINAL PROPERTIES**
  - PART NUMBER
  - FAMILY
  - GENDER
  - FINISH

Click  **Submit**.

LOAD WIRES FROM SCHEMATIC															
Wire Import Information															
✓	From Side Connector			To Side Connector			Wire Name	Colour	TwistPitch	Gauge	Material	CSA	Outer Dia	Wire Partnumber	Harnes
	ID	Cavity	Short Name	ID	Cavity	Short Name									
<input checked="" type="checkbox"/>	X-005	2		X-003	4		W001	GREEN	19mm	1	FLRY-A	1	2.1	FLRY-A-17GA-05	HAR-FL
<input checked="" type="checkbox"/>	X-005	1		X-003	1		W002	RED	19mm	2	FLRY-A	2	2.8	FLRY-A-14GA-02	HAR-FL
<input checked="" type="checkbox"/>	X-003	3		SP-003	X		W003	RED	19mm	0.35	FLRY-A	0.35	1.3	FLRY-A-22GA-02	HAR-FL
<input checked="" type="checkbox"/>	X-004	1		X-003	5		W004	BROWN	19mm	1	FLRY-A	1	2.1	FLRY-A-17GA-01	HAR-FL
<input checked="" type="checkbox"/>	X-001	2		X-002	2		W005	BROWN	19mm	1	FLRY-A	1	2.1	FLRY-A-17GA-01	HAR-FL
<input checked="" type="checkbox"/>	X-002	1		SP-003	X		W006	RED	19mm	0.35	FLRY-A	0.35	1.3	FLRY-A-22GA-02	HAR-FL
<input checked="" type="checkbox"/>	X-001	1		SP-003	X		W007	RED	19mm	0.35	FLRY-A	0.35	1.3	FLRY-A-22GA-02	HAR-FL
<input checked="" type="checkbox"/>	X-002	2		X-004	1		W008	BROWN	19mm	1	FLRY-A	1	2.1	FLRY-A-17GA-01	HAR-FL

Mismatched Connector Information														
	From Side Connector			To Side Connector			Wire Name	Colour	TwistPitch	Gauge	Material	CSA	Outer Dia	W
	ID	Cavity	Short Name	ID	Cavity	Short Name								
Connector Mismatch	GND-001	1		BAT-001	1		WIRE001	BROWN	19mm	1.0	DEFAULT	1.00	0.5	
Connector Mismatch	SP-002	X		SP-001	X		WIRE002	RED	19mm	1.0	DEFAULT	1.00	0.5	

Fig 40.1 Load Wires from Schematic

Using the **Load Wires from Schematic Pane** you can review the wires which have been found with matching connectors in the **Harness Layout**. You can check/uncheck individual wires you wish to import. Click **Submit** to load the schematic wires into the Harness Layout.

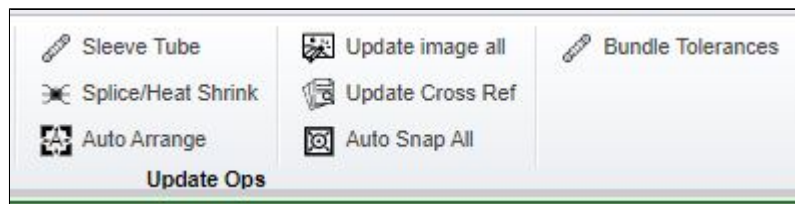



Fig 40.2. Update Ops in Harness Toolbar

To automatically match the DEFAULT wires you placed in schematic with wires that are available in the library use the  Wire PN Tool in the Harness toolbar:

UPDATE WIRE PART NUMBER			
Library:	<input type="text" value="quickstart"/>	Company:	<input type="text" value="CADONIX"/>
Search Combination	<input checked="" type="checkbox"/> CSA <input type="checkbox"/> OutsideDia <input type="checkbox"/> Gauge <input checked="" type="checkbox"/> Colour <input type="checkbox"/> Material	Approve Status	<input type="text" value="All"/>

Fig 40.3. Update Wire PN Form

Review the returned summary confirming which part numbers have been selected automatically in the rightmost column in **Fig 40.3**.

WIRE PART NUMBER UPDATES FROM COMPONENT DB									
	Sl.#	Wire Name	Gauge	Wire CSA	Outer Dia	Material	Colour	Multicore ID	Part Number
<input checked="" type="checkbox"/>	1	W004	1.0	1.00	2.10	FLRY	BR		WIRE_BR_1.0_FLRY
<input checked="" type="checkbox"/>	2	W005	1.0	1.00	2.10	FLRY	BR		WIRE_BR_1.0_FLRY
<input checked="" type="checkbox"/>	3	W008	1.0	1.00	2.10	FLRY	BR		WIRE_BR_1.0_FLRY
<input checked="" type="checkbox"/>	4	W001	1.0	1.00	2.10	FLRY	GN		WIRE_GN_1.0_FLRY
<input checked="" type="checkbox"/>	5	W003	0.35	0.35	1.40	FLRY	RD		WIRE_RD_0.35_FLRY
<input checked="" type="checkbox"/>	6	W007	0.35	0.35	1.40	FLRY	RD		WIRE_RD_0.35_FLRY
<input checked="" type="checkbox"/>	7	W002	2.0	2.00	2.80	FLRY	RD-GN		WIRE_RD_GN_2.0_FLRY
<input checked="" type="checkbox"/>	8	W006	2.0	2.00	2.80	FLRY	RD-GN		WIRE_RD_GN_2.0_FLRY

Fig 40.4. Selected Wire PN Summary

Finally close the wire Update Summary report returned which confirms the applied updates.

WIRE SUMMARY DETAILS									
Updated Wire Part Number(s)									
Wire Name	Gauge	Wire CSA	Outer Dia	Material	Primary Colour	Secondary Colour	Tertiary Colour	Multicore ID	Part Number
WIRE002	0.35	0.35	1.40	FLRY	RED	GREEN	NONE		WIRE_RD/GN_0.35_FLRY
WIRE006	1.0	1.00	2.10	FLRY	GREEN	NONE	NONE		WIRE_GN_1.0_FLRY
WIRE007	1.0	1.00	2.10	FLRY	BROWN	NONE	NONE		WIRE_BR_1.0_FLRY
WIRE008	2.0	2.00	2.80	FLRY	RED	GREEN	NONE		WIRE_RD/GN_2.0_FLRY
WIRE019	0.35	0.35	1.40	FLRY	RED	GREEN	NONE		WIRE_RD/GN_0.35_FLRY
WIRE020	0.35	0.35	1.40	FLRY	RED	GREEN	NONE		WIRE_RD/GN_0.35_FLRY
WIRE021	1.0	1.00	2.10	FLRY	BROWN	NONE	NONE		WIRE_BR_1.0_FLRY
WIRE022	1.0	1.00	2.10	FLRY	BROWN	NONE	NONE		WIRE_BR_1.0_FLRY


UnMatched Wire(s)

No Wires Found

Close

Fig 40.5. Updated Wire Summary

The Harness is updated. Notice the **Connectors** and the **Wire Table** located at the top left of the worksheet automatically update.

To automatically select all terminals and seals for connector in your harness use the  Cavities Tool from the Harness toolbar.


→ **Tip:** You can use the **Remove Wires**  command to remove all wires in the Harness prior to loading wires from a schematic.

---

## 41. Update the Schematic

Now that wires have been added to the Harness Layout, wire lengths have been automatically calculated as per the **Wire Table**. Using Arcadia it is possible to update the Schematic with these wire lengths in order to run Simulation on the schematic using the physical wire lengths.



Click the **Update Schematic**  command. The **Export Harness Wire Properties Pane** appears.

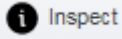
From the drop-down menus, set the **Schematic** to "**Fog Lamp Circuit**".

Click **Submit**.

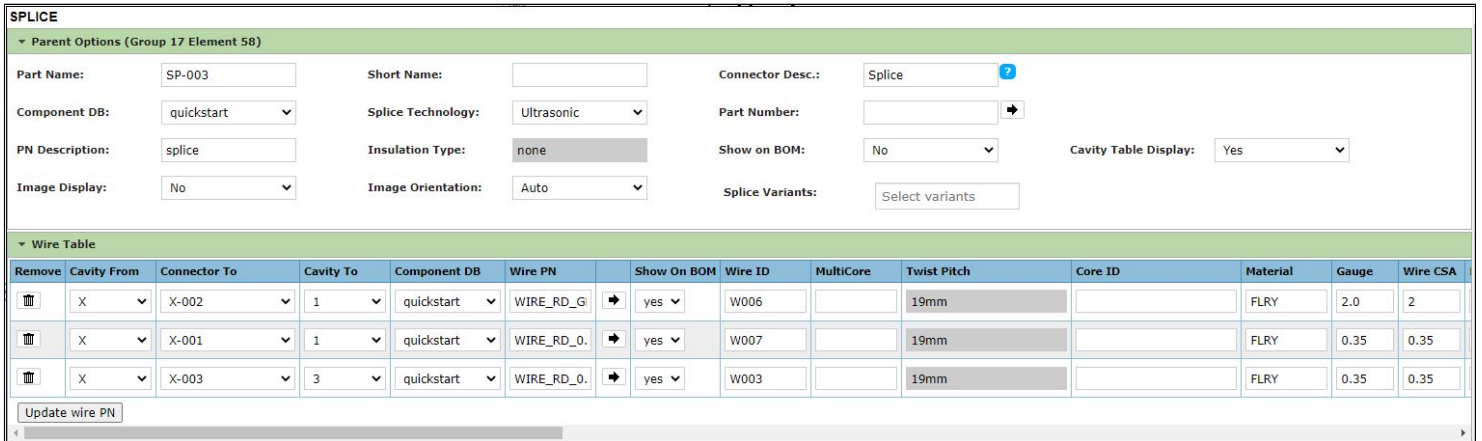
From the next screen are able to check / uncheck which wires you would like to update in the schematic. Click **Submit** to confirm the wire update.

## 42. Insert a Splice

You will now add the splice you created within the Component Database section,

If you Navigate/Find to SP-003 in the harness & Inspect  the Splice, a new Dialog box will appear, Here you can see the the Splice Technology has already been entered as Mechanical (Use the drop down menu if you wish to change this).

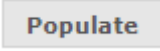
If you search a Part number **2140016** this will be filtered by this Splice Technology (**Fig 42.2**)

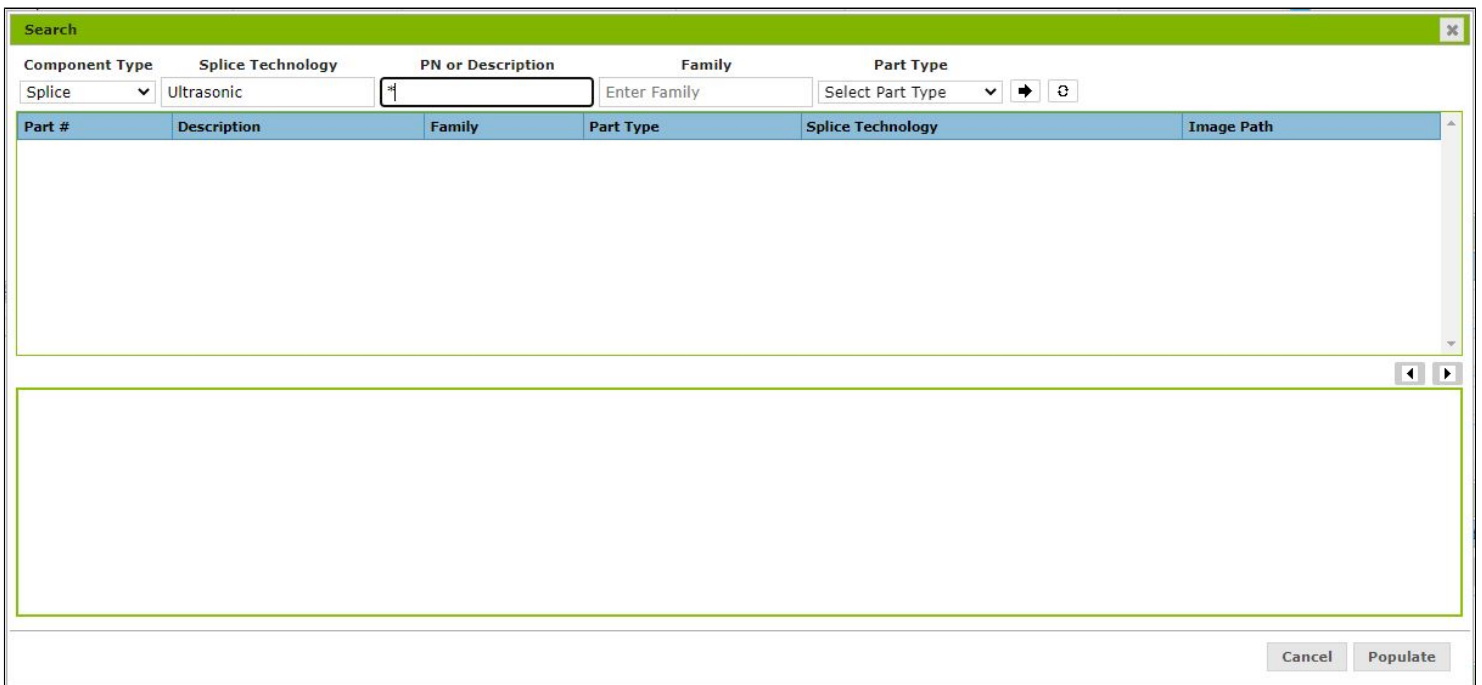


The screenshot shows the 'SPlice' dialog box. The 'Parent Options (Group 17 Element 58)' section includes fields for Part Name (SP-003), Short Name, Connector Desc. (Splice), Component DB (quickstart), Splice Technology (Ultrasonic), Part Number, PN Description (splice), Insulation Type (none), Show on BOM (No), Image Display (No), Image Orientation (Auto), and Splice Variants (Select variants). The 'Wire Table' section contains a table with columns: Remove, Cavity From, Connector To, Cavity To, Component DB, Wire PN, Show On BOM, Wire ID, MultiCore, Twist Pitch, Core ID, Material, Gauge, and Wire CSA. The table lists three wires with their respective properties.

Remove	Cavity From	Connector To	Cavity To	Component DB	Wire PN	Show On BOM	Wire ID	MultiCore	Twist Pitch	Core ID	Material	Gauge	Wire CSA
	X	X-002	1	quickstart	WIRE_RD_G	yes	W006		19mm		FLRY	2.0	2
	X	X-001	1	quickstart	WIRE_RD_0.	yes	W007		19mm		FLRY	0.35	0.35
	X	X-003	3	quickstart	WIRE_RD_0.	yes	W003		19mm		FLRY	0.35	0.35


Fig 42.1 Splice Dialog menu

Now you can click **Populate**  to select this Splice.



The screenshot shows the 'Search' dialog box. It has a search bar with 'PN or Description' set to '2140016'. The 'Component Type' is 'Splice' and the 'Splice Technology' is 'Ultrasonic'. The 'Family' field is empty, and the 'Part Type' is 'Select Part Type'. Below the search bar is a table with columns: Part #, Description, Family, Part Type, Splice Technology, and Image Path. The table is currently empty. At the bottom right, there are 'Cancel' and 'Populate' buttons.

Fig 42.2 Search

We can now run the **Wire Route**  command in order to change the wire orientation within the Splice & Finally Right click the Splice to Toggle Splice Image (**Fig 41.3**)

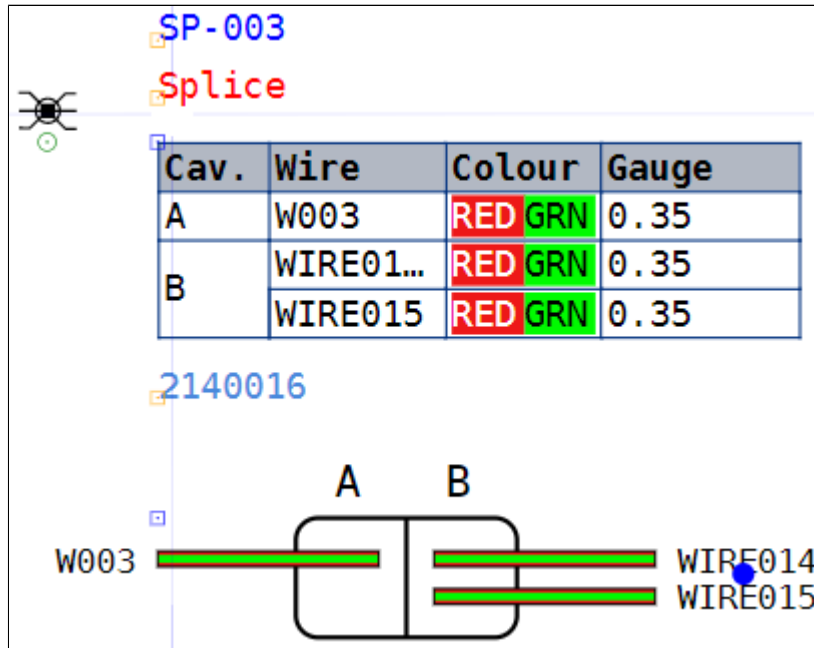


Fig 42.3. Toggling on the Splice Image is a key visual aid.

➔ **Tip:** You can select the Default Splice Technology from the first point of creating your harness, This means Arcadia will configure your splices this way from the get go

### Modify Harness

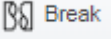
Information Management Performance Settings

<b>Work task</b> Quick Start Guide Harness	<b>Title</b> Fog Lamp Harness Complete	<b>Description</b> The Completed Harness
<b>Part Number</b> HAR-FL0001	<b>Revision</b> A1	<b>Drawn By</b> admin
<b>Drawn Date</b> 2015-07-04	<b>Profile</b> quickstart	<b>Editable Status</b> Editable
<b>Component DB</b> quickstart	<b>Units</b> mm - Millimeter	<b>Customer Part Number</b> 
<b>Customer Name</b> 	<b>Issue</b> 	<b>Checked By</b> user1
<b>Frame</b> A3 Landscape Standard	<b>Project</b> Quick Start Guide Updation	<b>Connector View</b> Generated
<b>Wire Names</b> Common Colors	<b>Default Splice Technology</b> 	<b>Table Type</b> HTML Table
<b>Approved By</b> user1	<b>Variant Group</b> Default	<b>Update Variant Group</b> No

Submit Cancel

Fig 42.4 Select the Default Splice Technology when inputting the Harness Information.

### 43. Insert Bundle Break

Next you are going to insert a **Bundle Break**  to the Harness, We can then move the bundle away from the harness but keep the intelligence of the wires and any lengths that have been applied to the bundle.

If you Navigate/Find the Harness back to the Connector X-001, You are going to insert the bundle break anywhere you choose on the bundle.

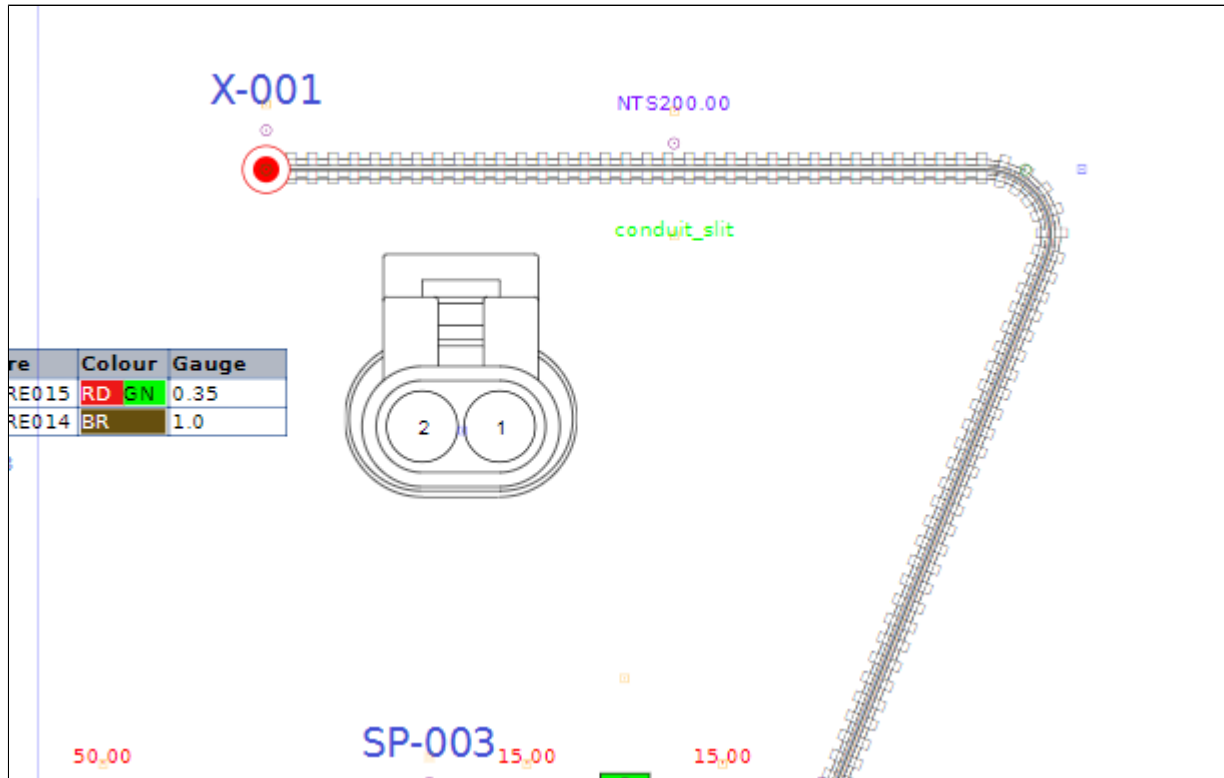
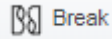


Fig 43.1

To locate the **Bundle Break**  feature please head to the tool bar and find this function under the heading "Bundle Ops".

Once we have selected the function we can place the **Bundle Break** (Fig 43.2).

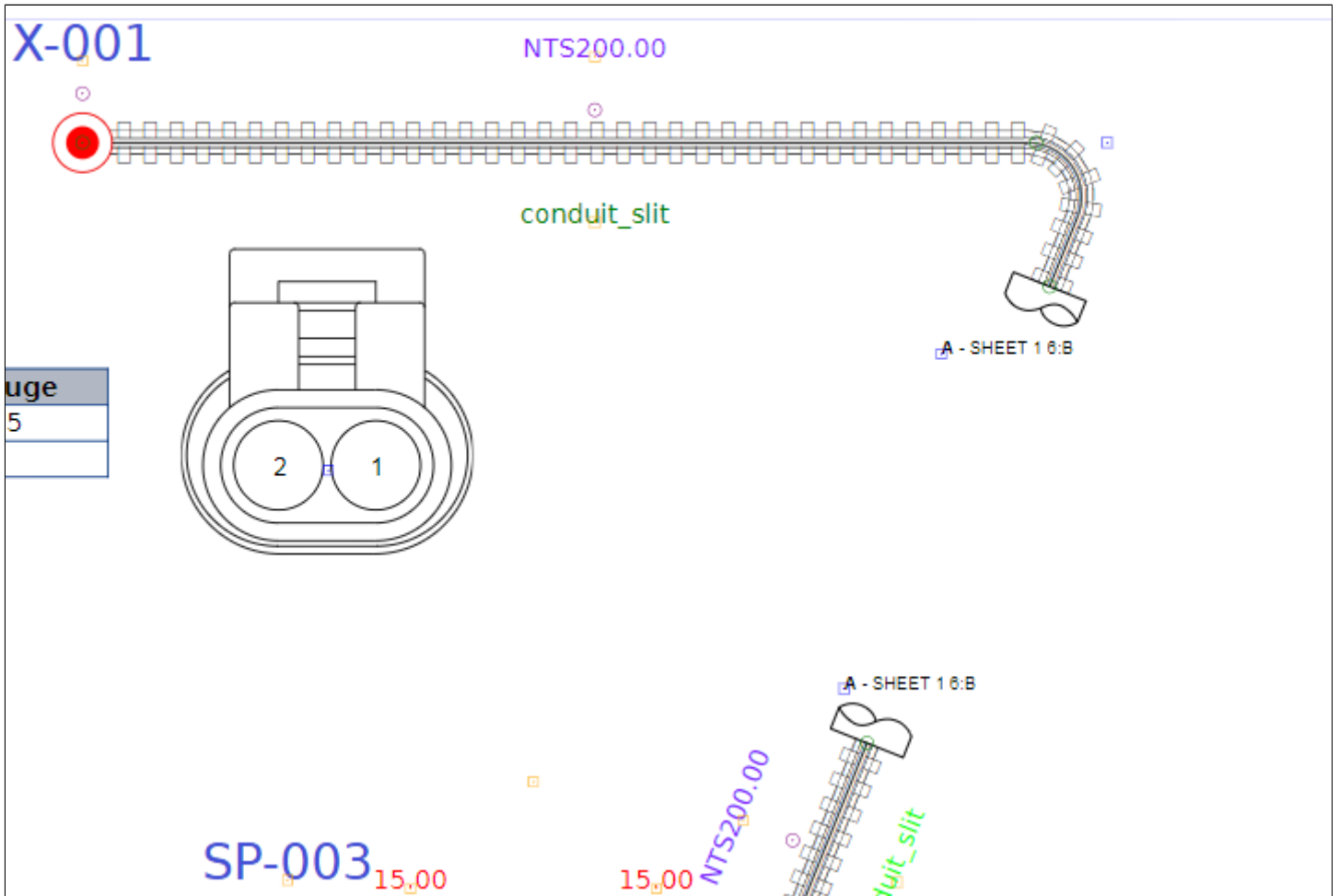


Fig 43.2 The break will reference the Zone for ease of location.

When Right Clicking the Bundle Break we can see a list of options, We can choose to Go to matching break which will zoom to the specified area of the other side of the break to help short cut any panning the harness screen.



Fig 43.3

## 44. Add a Dimension

You will now add a dimension to the harness.

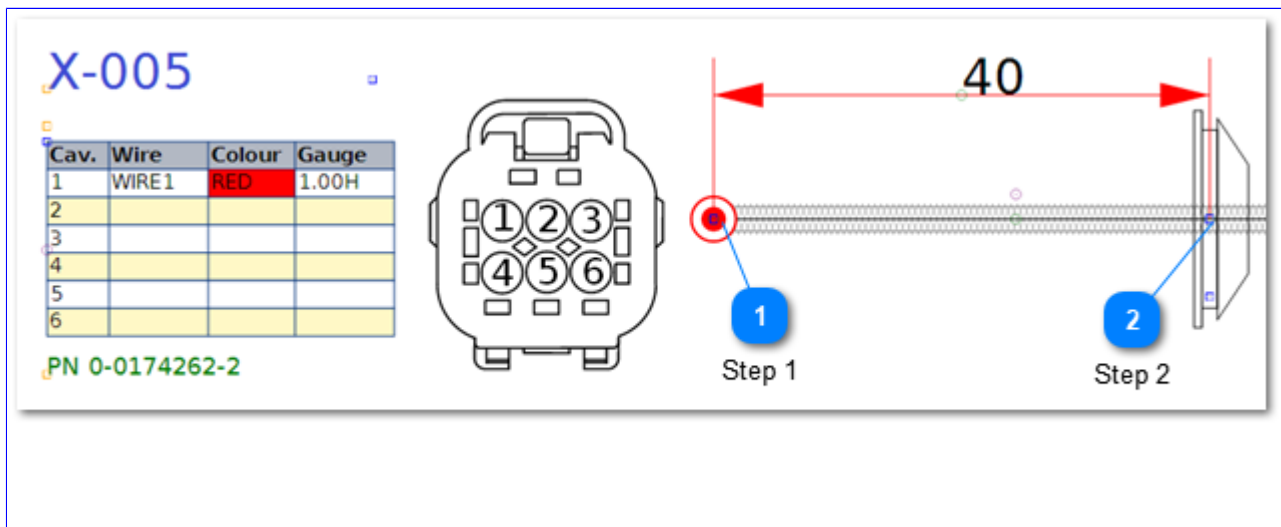



Fig 44.1 The Harness Window

### 1 Step 1

Click on the **Dimension** command  (Under the Drawing field). Then click on the **Harness Node**.

### 2 Step 2

Complete by clicking on the **Grommet Control Node**.

**Note:** The 3rd (last) mouse click sets the location of the dimension size, by moving the cursor along the distance indicator to set its position.

## CHAIN DIMENSION

The chain dimensioning tool in the harness helps the user to view the total length made by bundles from one node to another. If the bundle is set with the bundle tolerance, then the lengths can be shown along with the tolerance.

The chain dimension tool is placed in the bundle ops tool bar as shown below :



Fig 44.2 Chain Dimension Toolbar

The user must click on the chain dimension button in the tool bar and then choose a start node, end node and the position where text needs placing.

The dimensions can be toggled between the horizontal and vertical using the right-click and the default orientation can be customized in profile settings.

## 45. Insert a Clip

You will now add a Clip to the Fog Lamp Harness.

→ **Tip:** The dimension to each bundle end will be displayed as you hover over the bundle insert point, place the clip in the middle with 25mm each side as shown in Fig 45.1.

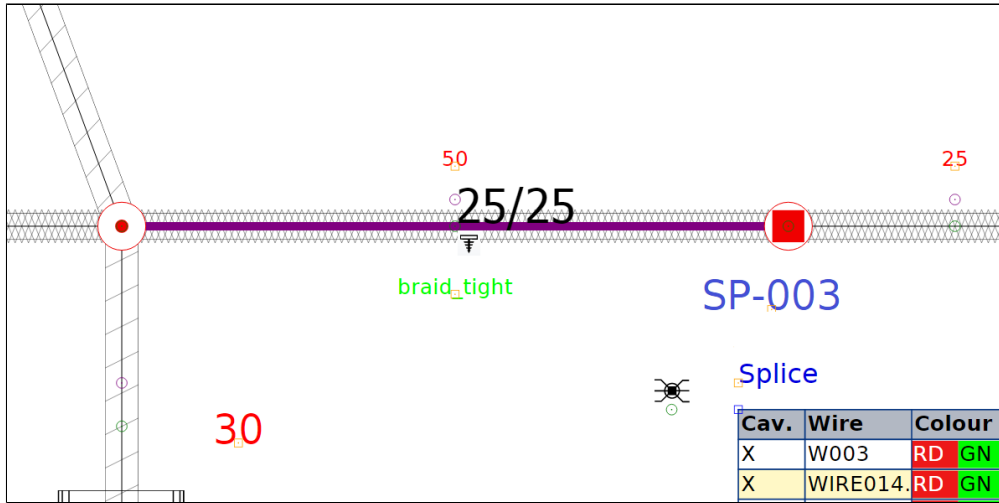


Fig 45.1 Insert a Clip



Add the clip by using the **Clip** command (On harness tab) and place it on the worksheet. Click on the bundle and location where you want to place the clip, then move the cursor in the orientation you want to place the clip, and press the left mouse button.

## 46. Annotate

You can add annotation to your Harness in Arcadia.

You will now add a **Green Tape** datum to the Harness.

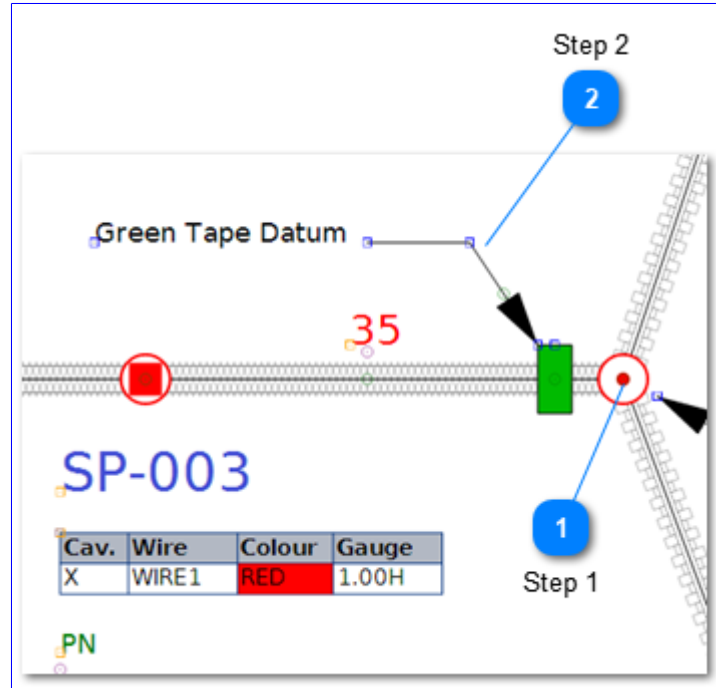




Fig 46.1 The Harness Window

### 1 Step 1

Click on the **Insert Tape**  command. Click to add the tape onto the harness and a node will automatically be added with the tape attached. Stretch out the tape to the desired size.

Use the **Inspect**  command to click on the **Tape Control Node**.

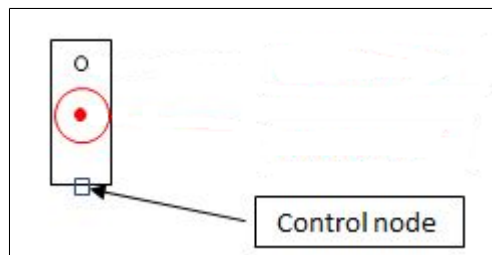


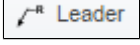
Fig 46.2 Control node location

Set the **Fill** value to **GREEN**. Click **Submit**.

You will now add some annotation text to your Harness.

### 2 Step 2

---

Click on the **Leader**  command (Under Drawing) and draw an arrow for the Green Tape Datum as shown. start by left clicking on the top of the tape then right click when to complete and add the information.

Enter the **Text** to read **Green Tape Datum**. Set the **Text Size** to **2.0** (under Advance Parent Options) and click **Submit**.

## 47. Bundle Tolerance

The bundle tolerances will have the set of values predefined in the Profile. Global tolerance can be set for the bundles in the profile.

This can be later shown as a text label along with the bundle length in the bundle using the label visibility function.

The bundle tolerance can be globally updated to all the bundles in one go with in the harness and there is also an option to update the tolerance per bundle in the bundle properties side pop

A new bundle tolerance icon is introduced in the harness tool bar as shown below.

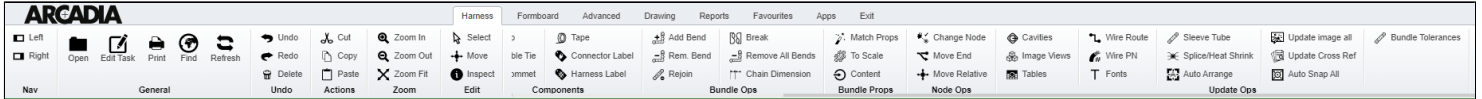


Fig 47 Bundle Tolerance

The user needs to click on the icon to see the side pop up where in they can configure the tolerances and the length ranges for that particular task.

	From Bundle Length (mm)	To Bundle Length (mm)	Lower Tolerance (mm)	Upper Tolerance (mm)
Length Range 1	0	3000	0	5
Length Range 2	3001	6000	0	10
Length Range 3	6001	Infinity	0	15

Alternatively, if the user wants to change the tolerance value for the particular bundle , then they can inspect the bundle to modify the tolerances as shown below

**BUNDLE**

13\_90C - 3810-1300  
SP-007  
Splice

Cav.	Wire	Colour	Ga
	WIRE019	BK	0.1
	WIRE020	BK	0.1
	WIRE023	BK	0.1
	WIRE025	BK	1.1
	WIRE085	BK	1.1

WIRE019  
WIRE020  
WIRE023  
WIRE025  
WIRE085

X

A B

WIRE019  
WIRE020  
WIRE023  
WIRE025  
WIRE085

X-101  
Battery POS

Cav.	Wire	Colour	Ga
1	WIRE056	RD	1.1
	WIRE061	RD	1.1

MS RING

**Parent Options (Group 217 Element 241)**

Part Name:  Set all Radius:

Drawn Length: 36.25mm(0mm setback) Override all Bend

Radius:

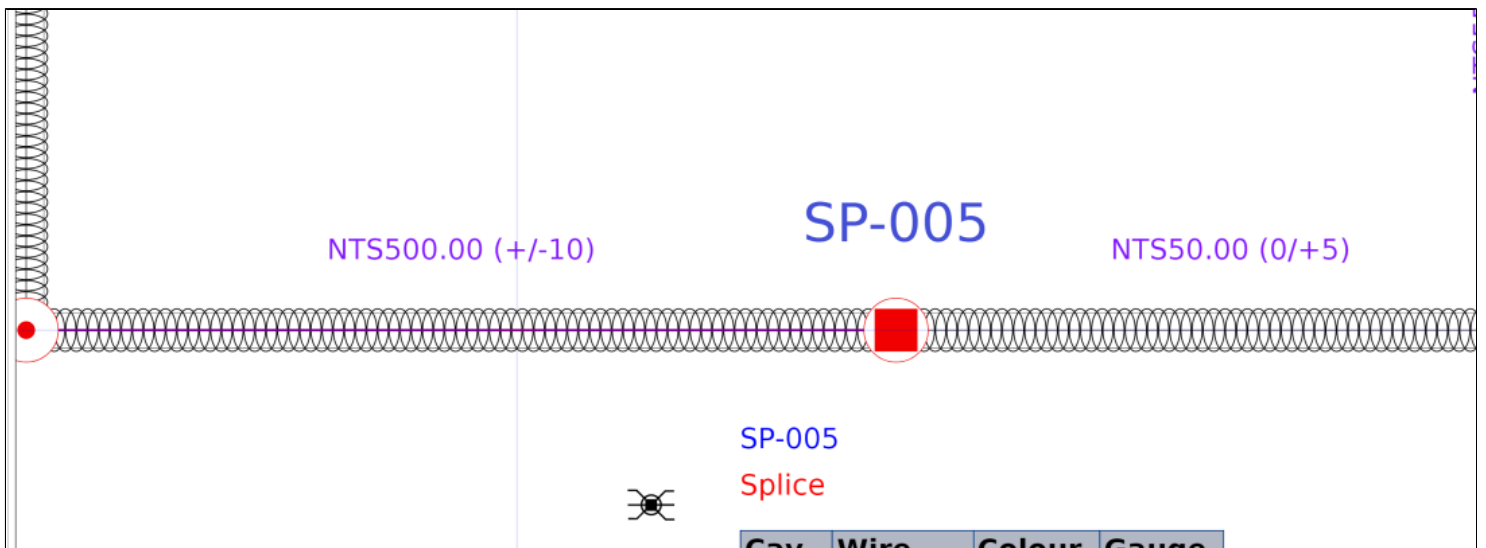
Enable Variant:  Wire Bundle Diameter: 1.53mm

**Set Length by Variants**

Variants:  Set Length:

Lower Tolerance:  Upper Tolerance:

Wire Bundle Diameter: 1.53mm



## 48. Clocking Angle Tool

The clocking angle tool can be used for a multitude of components including Connectors, clips and Bundle Branches.

You will now add a Clocking Angle to one of the connectors. To add a Clocking Angle click on the **Clocking Angle** command (Harness Tab) and click on connector X-002's control node to place a **clocking angle** for the connector (**Fig 48.1**). You can right click and switch between different representation of the clocking angle before left clicking to place it.

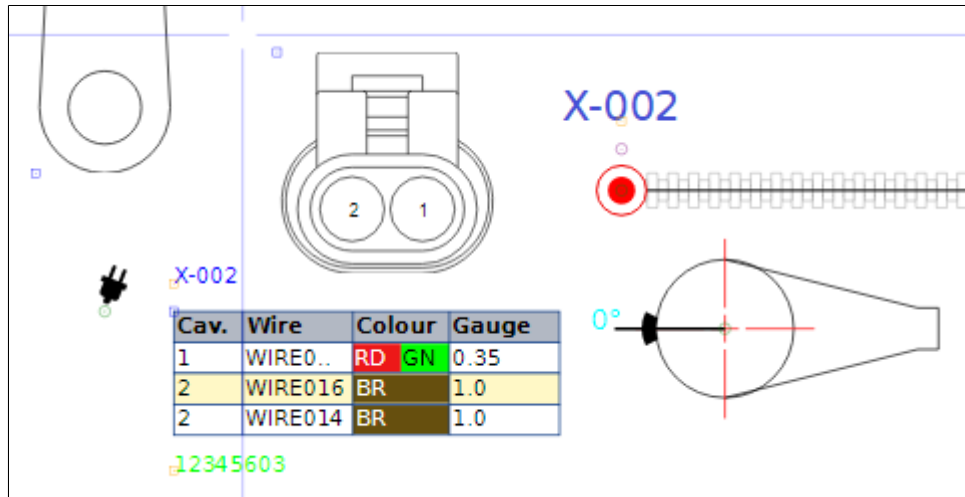


Fig 48.1. Placed Clocking angle tool

Set the in the Clocking Angle Properties windows set the Keyway Angle to 30° (this setting makes the connector pairs unique so they cannot be connected to the incorrect mating half for Military and Motorsport Harnesses). Set the Backshell angle to 90° this adds a 90° L bend to the connector (**Fig 48.2**). Click submit to add the Clocking angle to the Harness Drawing (**Fig 48.3**).

Stroke width: 0.1

Enter Text Here

Symbol Type: Backshell (90°)

Display Type: Angle

Set Keyway Angle: 0 360

Keyway Angle: 30

Set Backshell Angle: 0 360

Backshell Angle: 90

Submit Cancel

Fig 48.2. Clocking angle properties

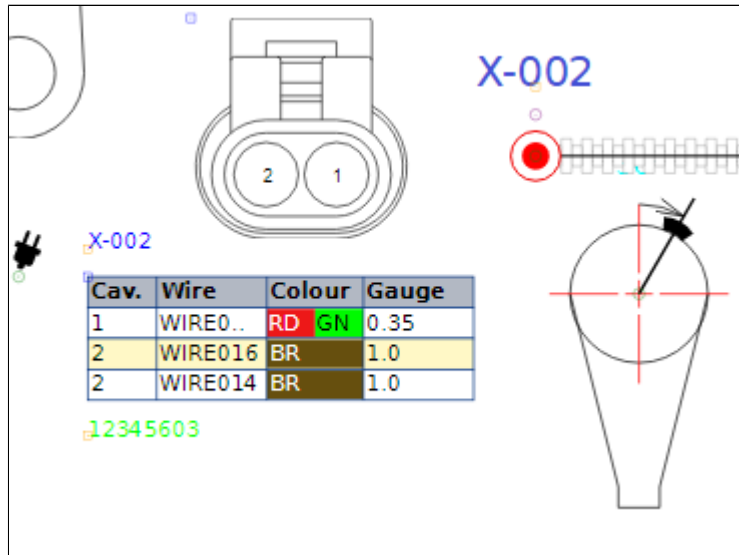




Fig 48.3. Updated clocking angle representation

## 49. Connector Label Tool

We will next add labels to all our connectors in our Harness using the **Connector Label Tool**  Connector Label

To add connector Labels to all connectors within your harness, Select the **Harness Tab, Connector label**


 Connector Label , Add an offset (Distance of the label from behind the connector) *Fig 49.1.*

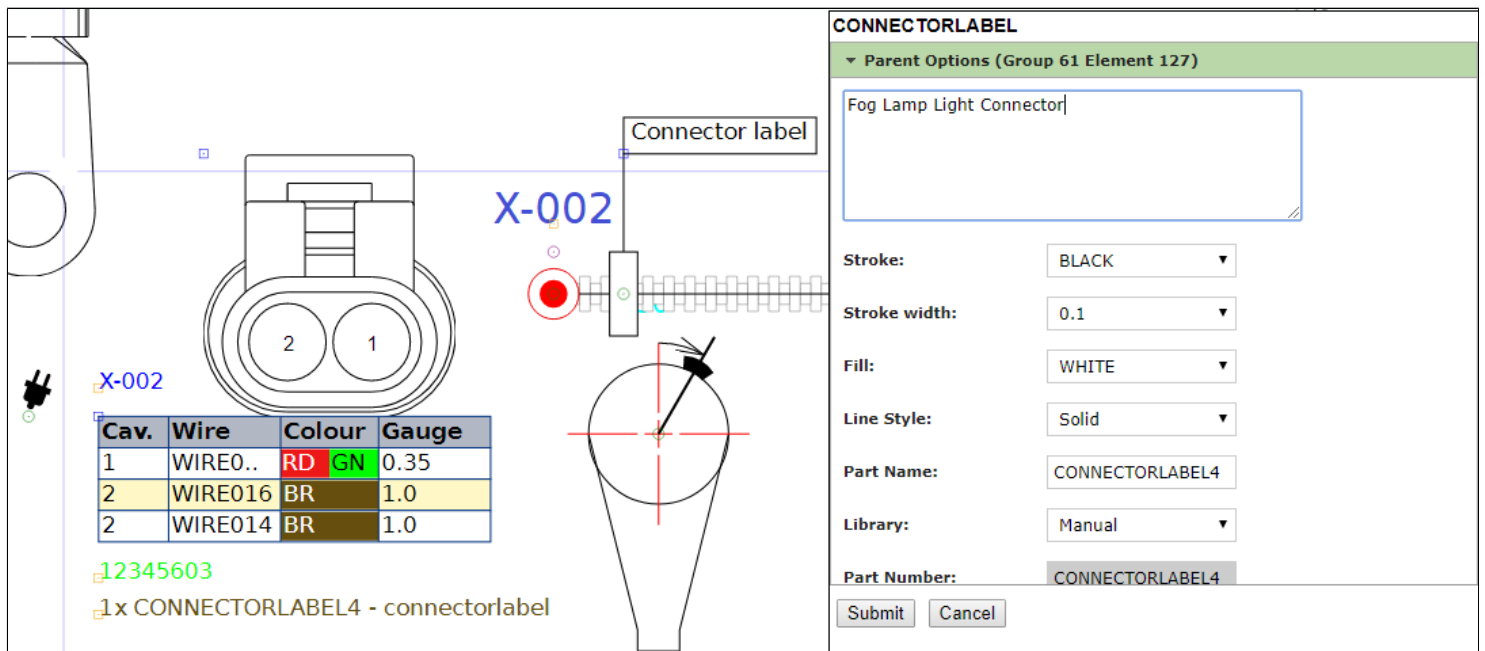


The dialog box titled "Place Connector Label" has a close button (X) in the top right corner. It contains a label "Label offset :" followed by a text input field containing "5mm". At the bottom left, there is a "Submit" button.

*Fig 49.1. Place connector label offset*

Labels can be moved (along with their leader lines) by Clicking the **Move tool**  Move and then the control node

for the Label, Labels can be edited by Clicking the **Inspect tool**  Inspect and the Labels control node



The image shows a software interface for editing a connector label. On the left, there is a technical drawing of a connector with a label "X-002" and a table of wire specifications. On the right, the "CONNECTORLABEL" properties dialog box is open, showing the label text "Fog Lamp Light Connector" and various styling options.

Cav.	Wire	Colour	Gauge
1	WIRE0..	RD GN	0.35
2	WIRE016	BR	1.0
2	WIRE014	BR	1.0

12345603  
1x CONNECTORLABEL4 - connectorlabel


**CONNECTORLABEL**  
Parent Options (Group 61 Element 127)  
Fog Lamp Light Connector

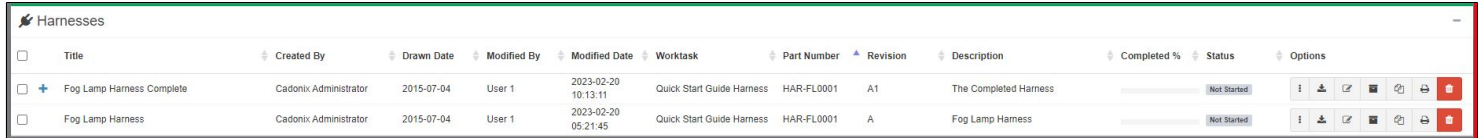
Stroke: BLACK  
Stroke width: 0.1  
Fill: WHITE  
Line Style: Solid  
Part Name: CONNECTORLABEL4  
Library: Manual  
Part Number: CONNECTORLABEL4  
Submit Cancel

*Fig 49.2. Connector Label properties*

\* Note The labels content will be built as per the profile setup for connector labels (*Fig 49.2*) details the connector name followed by the connector function if specified.

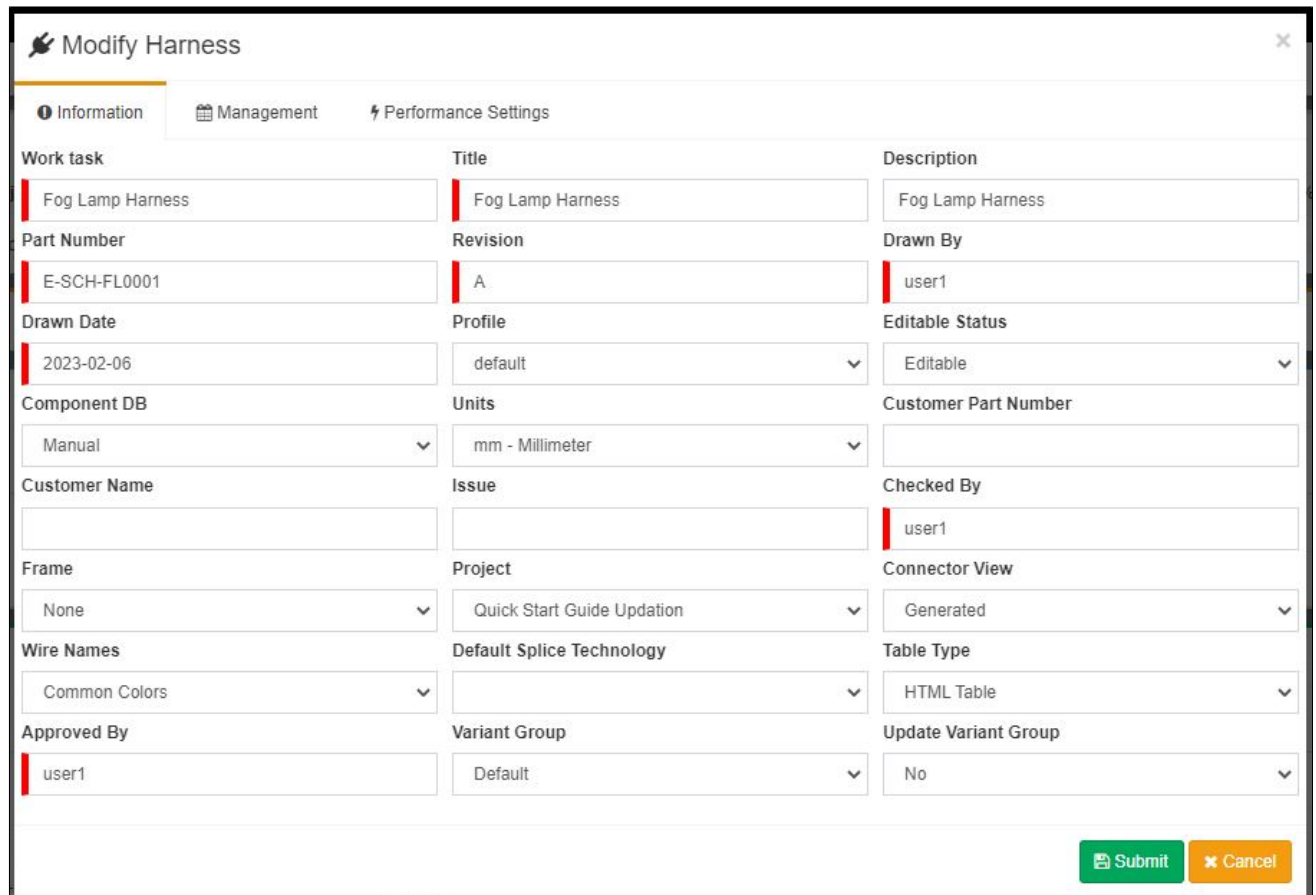
## 50. Generated Views with Wire Highlighting

Next we will add Spot highlighted connector views to our connectors these display wire colours, cavity location, multipoint crimps. To enable the **Generated Views** option for a drawing Click the Edit Task  icon under the Drawing selection screen (*Fig 50.1*) and setting **Connector** to **Generated** and **WireNames** to **Always** (*Fig 50.2*) and Submit to complete.



	Title	Created By	Drawn Date	Modified By	Modified Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
<input type="checkbox"/>	Fog Lamp Harness Complete	Cadonix Administrator	2015-07-04	User 1	2023-02-20 10:13:11	Quick Start Guide Harness	HAR-FL0001	A1	The Completed Harness		Not Started	
<input type="checkbox"/>	Fog Lamp Harness	Cadonix Administrator	2015-07-04	User 1	2023-02-20 05:21:45	Quick Start Guide Harness	HAR-FL0001	A	Fog Lamp Harness		Not Started	

Fig 50.1. Project Details page showing Harness Task details



Modify Harness		
Information	Management	Performance Settings
Work task	Title	Description
Fog Lamp Harness	Fog Lamp Harness	Fog Lamp Harness
Part Number	Revision	Drawn By
E-SCH-FL0001	A	user1
Drawn Date	Profile	Editable Status
2023-02-06	default	Editable
Component DB	Units	Customer Part Number
Manual	mm - Millimeter	
Customer Name	Issue	Checked By
		user1
Frame	Project	Connector View
None	Quick Start Guide Updation	Generated
Wire Names	Default Splice Technology	Table Type
Common Colors		HTML Table
Approved By	Variant Group	Update Variant Group
user1	Default	No

Fig 50.2. Modify Harness page showing Harness Task options

Fig 1, Shows the Details of connector X-003 on the Harness drawing with the spot Highlighter set, All cavities now show the wire colour inserted in them along with a leader line displaying the circuit/Wire its connected to i.e, Cavity 5, Brown W004.

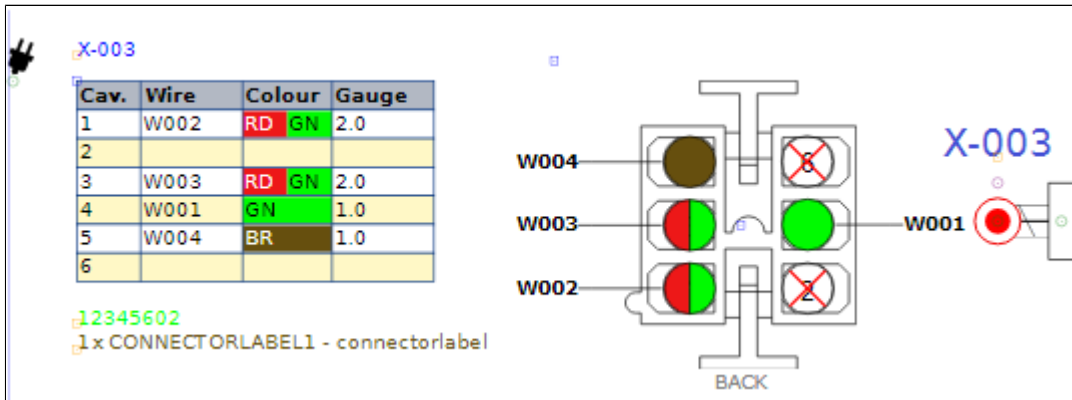


Fig 1 Connector X-003 Generated view with wire colours highlighted

Leader labels can be moved by right clicking the spot highlight control node and selecting **Move Wire Leads** (Fig 2) this can be useful for grouping wire colours together shown in Fig 3, to save the changes right click on the control node and select **Save**.

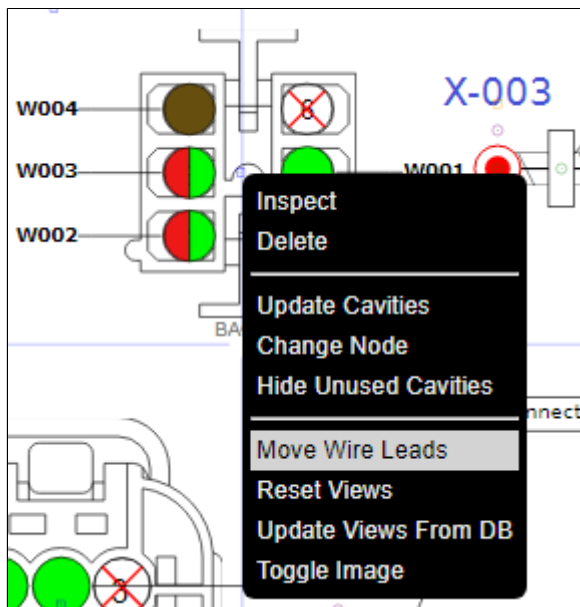


Fig 2 Options menu for spot highlighter

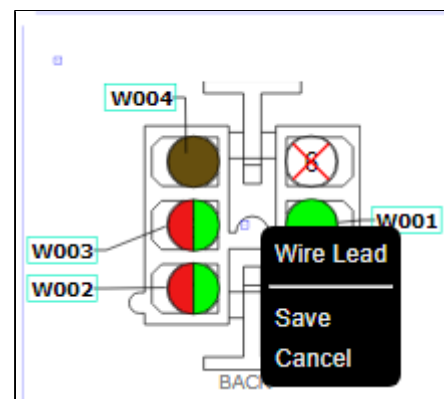
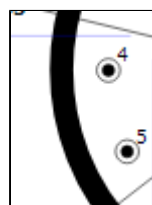
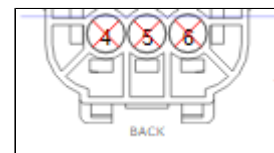


Fig 3 Leader move and colour grouping

Cavity options other than connected wires are displayed in as follows.



Cavity Plugs (None in Fog Circuit)



Empty Cavities

## 51. BOM Balloons

BOM balloons are circular callouts in the harness drawing that show bill of materials information for each component in the concerned harness.

BOM balloons usually show S.No /Item No ,corresponding to a Part Number in the BOM table. The BOM balloons can be placed only after placing a BOM table.

Types of BOM balloons:

- **Simple Balloons:** It will show only S.No inside the circle
- **Quantity Balloons:** The circle will be split to show the S.No in the upper half and the quantity of the part number in the lower half of the circle

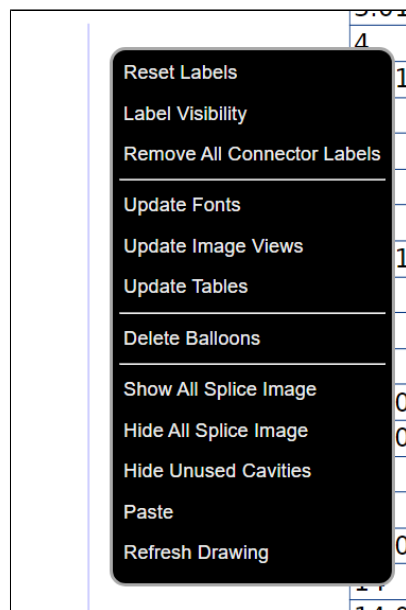
Right click on the BOM table and click on the “Update BOM” button to update the balloons with the latest BOM data. The balloon properties can be configured in the profile:

Wire CSA	From Tag	To Tag
1.00	X-005	X-003
2.00	X-005	X-003
0.35	X-003	SP-003
1.00	X-004	X-003
1.00	X-001	X-002
2.00	X-002	SP-003
0.35	X-001	SP-003
1.00	X-002	X-004

S.No	Quantity	Length (mm)	Part No.	Part Type
1	2		12345603	Connector
2	1		12345602	Connector
3	1		12345601	Connector
4	1		174263-7	Harness node attach part
5	1		1-174263-1	Harness node attach part
6	1		GROMMET1	Harness node attach part
7	1		TAPE1	Harness node attach part
8	2		12345619	Terminal
9	1		61117-1	Terminal
10	1		12345621	Terminal
11	1		12345620	Terminal
12	1		171662-1	Terminal
13	2		350557-1	Terminal
14	1		350078-1	Terminal
15	1		12345614	Seal
16	1		172746-1	Seal
17	4		172746-1	Other Part
18		4651	WIRE BR_1_0_FLY	Wire
19		601	WIRE RD_0_35_FLY	Wire
20		2442	WIRE RD_GN_2_0_FLY	Wire
21		379	WIRE_GN_1_0_FLY	Wire

Fig 51 BOM Balloons

If the users want to delete a balloon, they can right click on the balloon to delete a single balloon or right click on the empty space to get a context menu and click on “Delete Balloons”



## 52. Traffic Light System

The traffic light status will be on the advanced toolbar in Harness, that change it status to RED, AMBER or GREEN as we do the design. Clicking this Icon will load the below checks & the status on the right-side POP-UP.

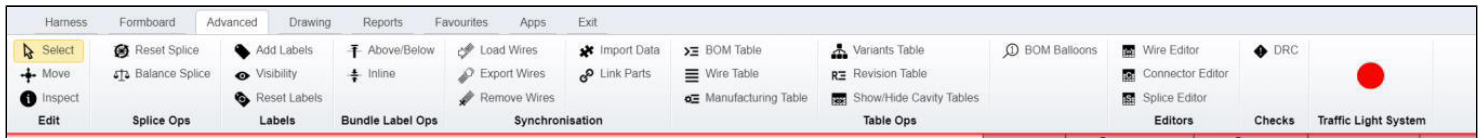


Fig 52 Traffic Light System

TRAFFIC LIGHT SYSTEM SUMMARY		
Function	Status*	Remarks
Design Rule Check (DRC)		DRC may not be Up-to-date
Wire Route		
Sync from Schematic		

Fig 52.1 Traffic Light System Summary

There will be a set of new macros introduced that can be used within the Harness drawing/frames to give the status in the drawing.

- **Schematic Sync** - Show Schematic Name & Revision. This info is already saved. If any manual edit of wires happens in Harness, the macro should display "Manual".
- **Route Status** - If Wire route has been updated. Yes/No. Please note that if there is any operation performed on the harness that would change the wire route like insert node, insert bundle, the status will be updated appropriately
- **DRC Errors** - Count of the DRC errors/warnings

### 53. BOM Table Enhancement

The BOM table now has got now couple of option to group/sort the BOM table.

You can also override the profile settings at any point of time inside the harness, by inspecting the BOM table and changing the properties as required.

There is also Item No column where in you have you own number for the each line item of the component and this will be reflected in the Balloon as well.

Also, the reference column, will be populated with connector/splice reference code for the connector and splices respectively.

Please refer to the BOM table below, where the table is grouped by connector/splice and the item No./ reference is turned ON.

**BOM TABLE**

▼ Parent Options (Group 62 Element c78f1e9868ef11ea835b0242d255f5c4)

Part Name: <input type="text" value="BOMTABLE1"/>	Component DB: <input type="text" value="quickstart"/>
Part Number: <input type="text"/>	PN Description: <input type="text" value="bomtable"/>
Show on BOM: <input type="text" value="No"/>	Variant Type: <input type="text" value="ALL"/>
Part Category: <input type="text" value="None"/>	Table Type: <input type="text" value="HARNESS BOM"/>
BOM Table Header: <input type="text" value="Top down"/>	Show Multiple PN: <input type="text" value="No"/>
Show Multiple: <input type="text" value="No"/>	Group by Connector / Splice: <input type="text" value="No"/>
Company: <input type="text"/>	
Table Sort by: <input type="text" value="Comp type"/>	Table Group by: <input type="text" value="Part Number"/>

▼ Table Layout

Column Order	Column Name	Visible	Name as Displayed	Cell Width
↑ ↓	S.No	Yes ▼	<input type="text" value="S.No"/>	<input type="text" value="4.71"/>
↑ ↓	Item No	Yes ▼	<input type="text" value="Item No"/>	<input type="text" value="8"/>
↑ ↓	Reference	Yes ▼	<input type="text" value="Reference"/>	<input type="text" value="12"/>
↑ ↓	Quantity	Yes ▼	<input type="text" value="Quantity"/>	<input type="text" value="8.53"/>
↑ ↓	Length (mm)	Yes ▼	<input type="text" value="Length"/>	<input type="text" value="12.34"/>
↑ ↓	Part No.	Yes ▼	<input type="text" value="Part No."/>	<input type="text" value="19.01"/>
↑ ↓	Comp Type	Yes ▼	<input type="text" value="Part Type"/>	<input type="text" value="23.78"/>
↑ ↓	Description	Yes ▼	<input type="text" value="Description"/>	<input type="text" value="47.6"/>
↑ ↓	Supplier Name	Yes ▼	<input type="text" value="Supplier Name"/>	<input type="text" value="13.29"/>
↑ ↓	Supplier Part No.	Yes ▼	<input type="text" value="Supplier Part No."/>	<input type="text" value="19.01"/>
↑ ↓	Location	Yes ▼	<input type="text" value="Location"/>	<input type="text" value="11.38"/>
↑ ↓	Variant	Yes ▼	<input type="text" value="Variant"/>	<input type="text" value="7.57"/>

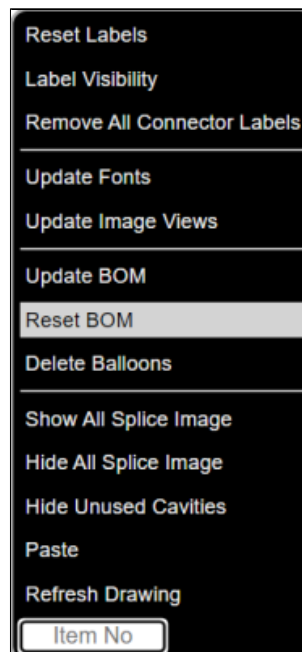
Fig 53 BOM Table

Once the BOM table is placed, any new component placed will be added to the last line item unless it is associated with the existing PN

If a component is deleted, the history will be maintained with a strike through as shown below.

25	Clip1	25	1		09363.00.087	Harness node attach part	FIXING CLIP	ITW AUTOMOTIVE	09363.00.087	7G(1)
26		26	1		9806115	Harness node attach part	T-PIECE 13-13-13_FORM B	SCHLEMMER	9806115	5D(1)
27		27	1		9876543	Harness node attach part	GROMMET ID 10MM	CADONIX	9876543	8C(1)
28		28	2		M5 RING TERMINAL	Terminal	M5 RING TERMINAL	CADONIX	M5 RING TERMINAL	4F(2)
29		29		695	3810-1300	Sleeve & Tube	SLIT CONDUIT NW 13_90C	Manual	3810-1300	
30		30		150	1967747	Sleeve & Tube	UNSLIT CONDUIT NW 7.5/UFW PP-BS	SCHLEMMER	1967747	
31		31		430	1967847	Sleeve & Tube	SLIT CONDUIT NW 7.5/UFW PP-BS	SCHLEMMER	1967847	
32		32		150	1967750	Sleeve & Tube	UNSLIT CONDUIT NW 10/UFW PP-BS	SCHLEMMER	1967750	
33		33		815	1967850	Sleeve & Tube	SLIT CONDUIT NW 10/UFW PP-BS	SCHLEMMER	1967850	
34		34		305	MC WIRE_1.0_SILICON_YE	Multicore		Manual	MC WIRE_1.0_SILICON_YE	
<del>35</del>		<del>35</del>		<del>1480</del>	<del>WIRE_RD_0.5_FLYR</del>	<del>Wire</del>	<del>FLRY-B_0.5mm2_105 C</del>	<del>CADONIX</del>	<del>WIRE_RD_0.5_FLYR</del>	
35		35		665	WIRE_RD_0.5_FLYR	Wire	FLRY-B_0.5mm2_105 C	CADONIX	WIRE_RD_0.5_FLYR	
36		36		300	WIRE_RD_1.0_FLYR	Wire	FLRY-B_1.0mm2_105 C	CADONIX	WIRE_RD_1.0_FLYR	
37		37		720	WIRE_RD/BK_0.75_FLYR	Wire	FLRY-B_0.75mm2_105 C	CADONIX	WIRE_RD/BK_0.75_FLYR	
38		38		160	WIRE_RD_2.0_FLYR	Wire	FLRY-B_2.0mm2_105 C	CADONIX	WIRE_RD_2.0_FLYR	
39		39		685	WIRE_YE/RD_1.5_FLYR	Wire	FLRY-B_1.5mm2_105 C	CADONIX	WIRE_YE/RD_1.5_FLYR	
40		40		1075	WIRE_BK_1.5_FLYR	Wire	FLRY-B_1.5mm2_105 C	CADONIX	WIRE_BK_1.5_FLYR	
41		41		690	WIRE_BK/WH_1.0_FLYR	Wire	FLRY-B_1.0mm2_105 C	CADONIX	WIRE_BK/WH_1.0_FLYR	
42		42		1460	WIRE_RD_1.5_FLYR	Wire	FLRY-B_1.5mm2_105 C	CADONIX	WIRE_RD_1.5_FLYR	

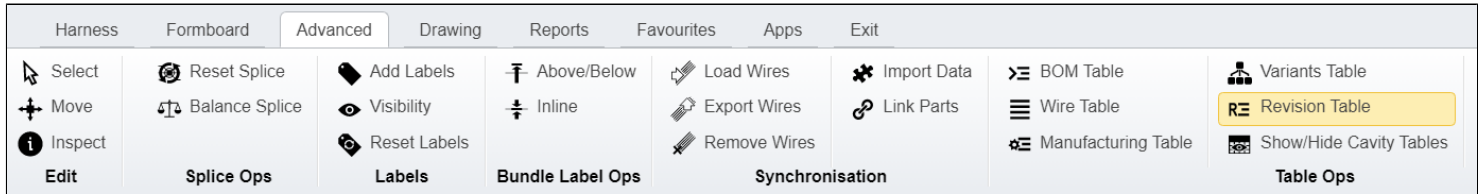
The user can always reset the BOM to default by right clicking on the table and clicking on “Reset BOM”



There will not any history maintained in the Formboard and every time the formboard is synchronized with the harness, the BOM will be reset to affect the latest changes.

## 54. Harness - Sheet Revision Table

The sheet revision table icon can be found in the advanced toolbar as shown below.



*Fig 54 Revision Table*

The macros can be modified to show only the modified date rather than both the date and time from the macros section.

Revision	Comments	Date and Time	Modified by
A	Terminal 824012-001 added to cvaity A of C-001	23/06/02 06:54:01	User1

## 55. DRC & Printing



Use the **Design Rule Check** command on the toolbar to perform number of checks to help ensure the harness is ready for manufacture. The checks are grouped into sections to make it easier to read - take a moment to look at the analysis at this stage. Warnings associated with terminal and seal selection are also displayed.

Summary			
⚠ WARNING			31
✖ ERROR			8
Part Number Validation Report			
Action	Name	Part Number	Message
⚠ WARNING	bundle2		No Part Number available for bundle2
⚠ WARNING	bundle3		No Part Number available for bundle3
⚠ WARNING	bundle5		No Part Number available for bundle5
⚠ WARNING	bundle6		No Part Number available for bundle6
⚠ WARNING	bundle4		No Part Number available for bundle4
⚠ WARNING	bundle1		No Part Number available for bundle1
⚠ WARNING	bundle8		No Part Number available for bundle8
⚠ WARNING	bundle7		No Part Number available for bundle7
⚠ WARNING	bundle9		No Part Number available for bundle9
⚠ WARNING	node14	<u>GROMMET1</u>	Node - attached part not found in the quickstart component db
⚠ WARNING	node15	<u>TAPE1</u>	Node - attached part not found in the quickstart component db
Wire Length Analysis Report			
Action	Name	Part Number	Message
✓	No Errors found		
Cavity Data Analysis Report			
Action	Name	Part Number	Message
✓	No Errors found		
Wire Data Analysis Report			
Action	Name	Part Number	Message
✓	No Errors found		
Wire Diameter Analysis Report			
Action	Name	Part Number	Message
✖ ERROR	W002	<u>WIRE_RD_GN_2.0_FLY</u>	Terminal: <u>61117-1</u> does not fit Wire Outer Diameter 2.8mm for the connector <u>12345602</u> in cavity 1
<div style="display: flex; justify-content: space-between; margin-top: 10px;"> <span>Close</span> <span>HTML Export</span> <span>Go to DRC Settings</span> </div>			

Fig 55 Design Rule Checks

Close down the Design Rule Check pane to head back to the fog lamp harness.

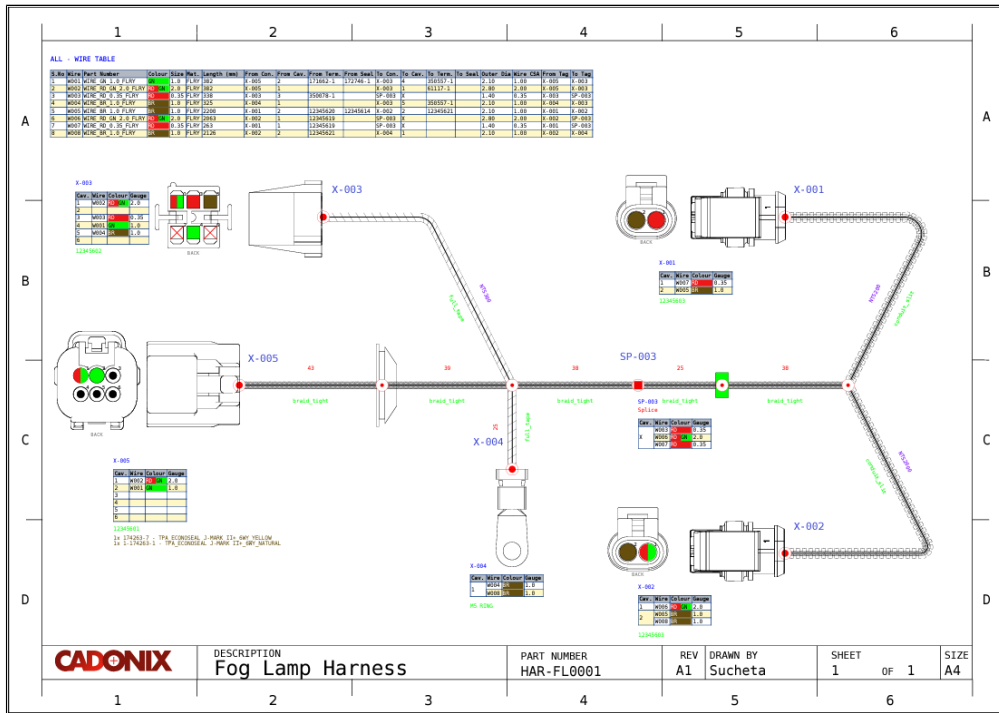


Fig 55.2 The Fog Lamp Harness



The **Print** command allows the Harness to be printed in colour or monotone. This will generate a new window with your schematic available as a PDF or to be printed directly. You can select to print the Entire sheet or Print by Zone, Also if you highlight a selected area you can print a specific area.

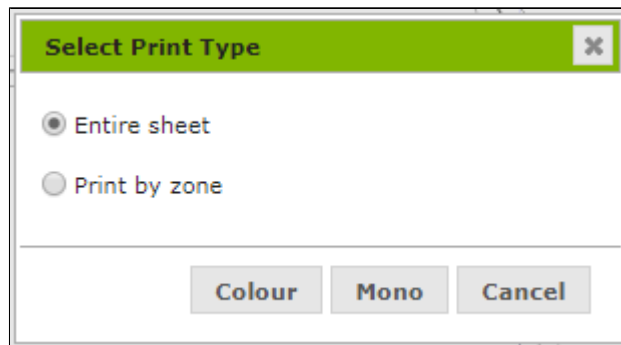


Fig 55.3 Print Window



## 56. Run Reports

**Note:** Reporting is not available on all versions of Arcadia and is a specific feature in the Manufacturing Professional module.

You can run a number of powerful reports in Arcadia. You can access them from the **Project Details** screen.

When working on a harness task you can get to the **Project Details Screen** by clicking on the **"Exit"** toolbar and selecting **"Exit Drawing"**.

Now on the **Project Details** screen, navigate to the **Harnesses** section and to the right of the **Fog Lamp Harness**

use the **Reports** icon  and then choose the .

On the right-hand-side are a selection of reports you can run including **Wire Cutting List**, **Harness Summary** and a complete **Bill of Materials**.

You can also edit the **Manufacturing Settings** to meet any Manufacturing requirements and edit the Design or Outputs of the reports.

From the **Home** menu (**Fig 56.1**) select **Settings** then **Company Profiles** (**Fig 56.2**) then **"Quickstart" Profile** (**Fig 56.3**) and next select **Reports** (**Fig 56.4**).

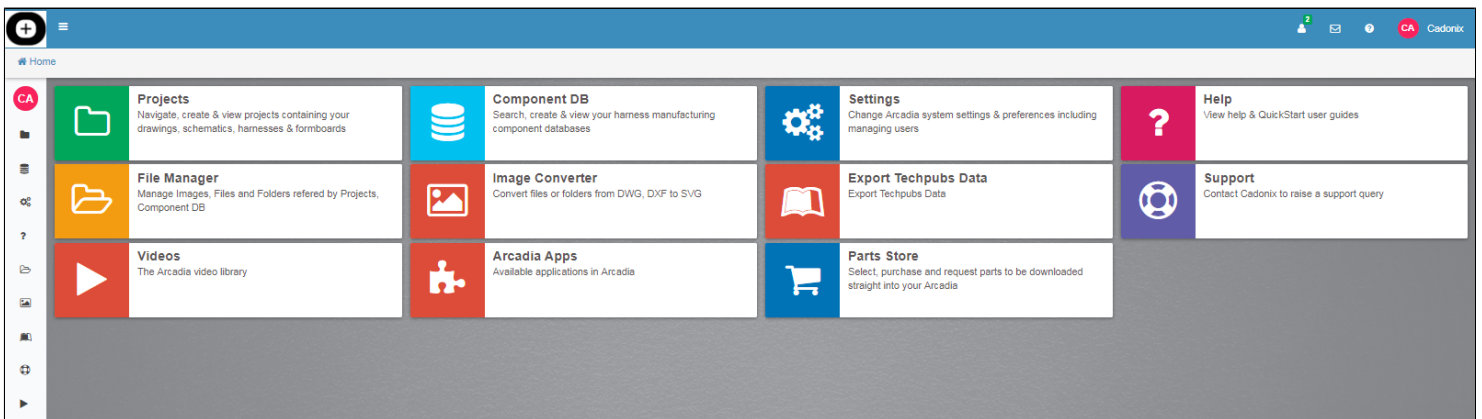


Fig 56.1 Home Menu

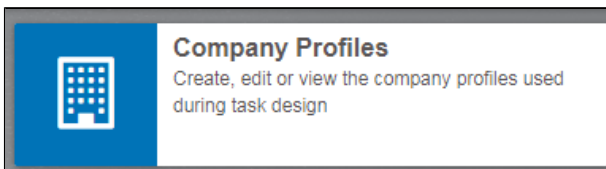


Fig 56.2 Click the Profile to Enter

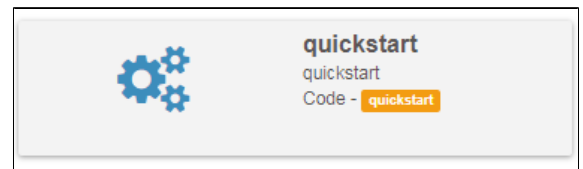


Fig 56.3 Click Quickstart to Edit the Quickstart Profile

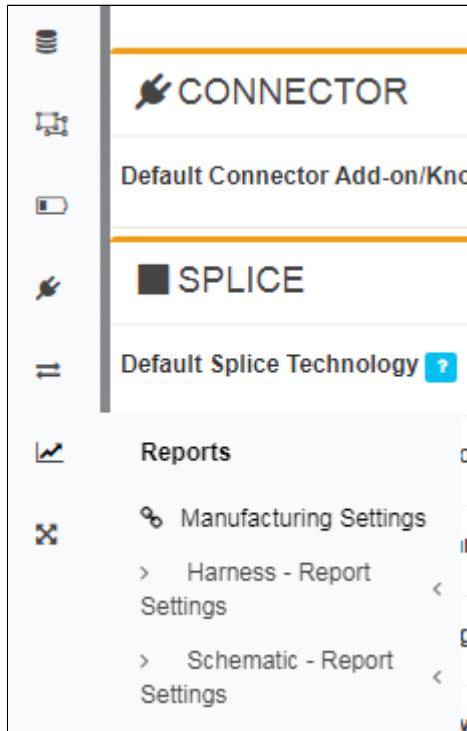
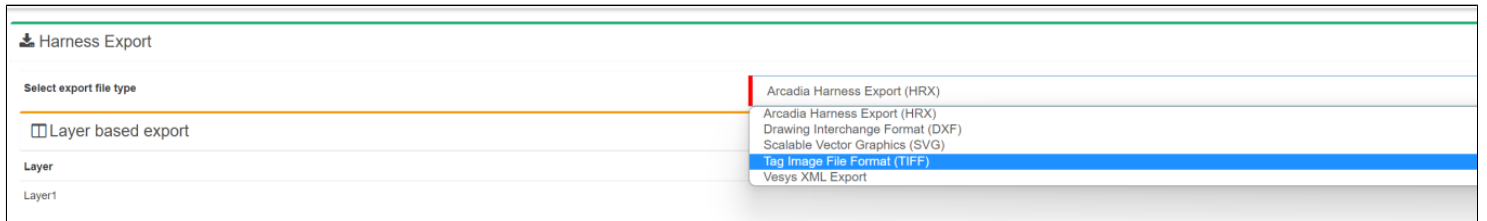


Fig 56.4 see a list of Manufacturing & Reports you can edit

## 57. Harness to TIFF Export

The Tiff export settings can be configured in the profile.

The users can export the TIFF from the harness export drop down as shown below.

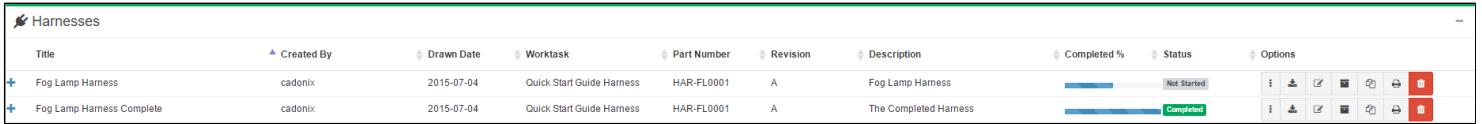


*Fig 57 Harness Export*

## 58. Formboard

Arcadia provides the ability to generate a full scaled harness or 'Formboard' layout which would typically be used by a harness manufacturer.



➔ **Note:** Formboard is an optional module and is only available in the Arcadia Manufacturing Professional edition.



Title	Created By	Drawn Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Harness	cadonix	2015-07-04	Quick Start Guide Harness	HAR-FL0001	A	Fog Lamp Harness	<div style="width: 50%;"></div>	Not Started	[Info] [Edit] [Print] [Share] [Delete]
Fog Lamp Harness Complete	cadonix	2015-07-04	Quick Start Guide Harness	HAR-FL0001	A	The Completed Harness	<div style="width: 100%;"></div>	Completed	[Info] [Edit] [Print] [Share] [Delete]

Fig 58.1 Project Details page showing Harness Task details

To generate a formboard harness, navigate back to the **Project Details** page.

Under the **Harnesses** section click on Option List  and select **Formboard** . A new task will be generated under the Harness Formboard section.

A dialogue box will come up saying 'Form Board Created Successfully' Click OK.

To make it easier to distinguish your Formboard from your Harness drawings you can Modify it's **Title** and **Description** (Fig 58.3) by clicking the Formboards **Edit Task** option (Fig 58.2) from the Harness Project details page and updating the Formboards details.

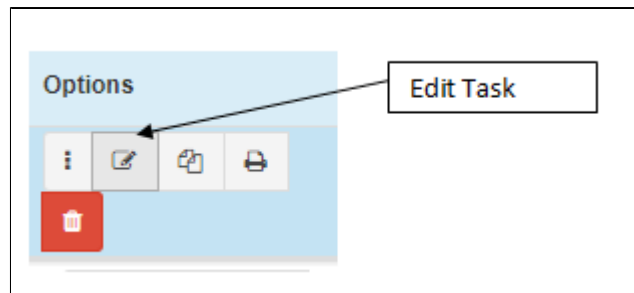


Fig 58.2 Options Edit Task

Fig 58.3 Formboard properties

Title	Created By	Drawn Date	Worktask	Part Number	Revision	Description	Completed %	Status	Options
Fog Lamp Harness	cadonix	2015-07-04	Quick Start Guide Harness	HAR-FL0001	A	Fog Lamp Harness		Not Started	[Icons]
Fog Lamp Harness	cadonix	2015-07-04	Quick Start Guide Harness	HAR-FL0001	A	Fog Lamp Harness		Not Started	[Icons]

Fig 58.4 Harness Formboard Task details.

Click on the associated task under **Harness Formboard**. Once opened you will notice the following changes to the the toolbar.

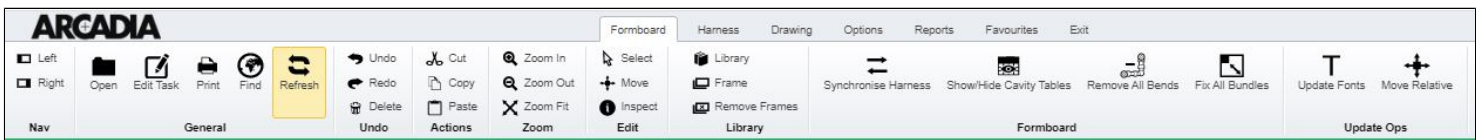




Fig 58.5 Formboard Toolbar Commands

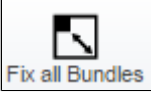
In Arcadia, a harness formboard is linked to the original harness task from which it was generated - this becomes the 'master' harness. As a result, any changes made to the master harness can be synchronised with the formboard - simplifying the upkeep of the formboard drawing.



As a result a number of commands are greyed out to prevent accidental changes to the harness which would break the master-formboard link. Instead it is possible to make changes to the master harness and then use the


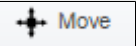
command **Synchronise Harness**  when in the formboard, which will bring across the updates.

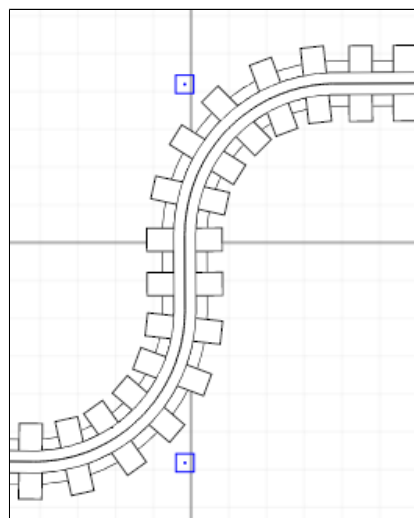
Change the **Drawing Frame**  and select the frame **A0 Landscape Plane** - which will be used for the purpose of laying out the formboard harness. Click submit.

Use the command **Fix Bundle Length** to convert a not-to-scale (NTS) bundle to a full scaled bundle. Click the

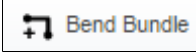
**Fix all Bundles** command  the server will process the request and then return you to the scaled formboard harness.

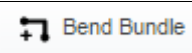
Now that the bundles are true to their length you can use **Add Bundle Bend**  (On Harness Tab) to add bends to the bundle which can be used in along with the **Move**  command to layout the harness to fit onto a formboard drawing frame.

To do this click the **Add Bundle Bend**  click on a point on the bundle you wish to bend, next click the **Move**  command and then on the Bend Bundle node (Small blue Square) as you move your mouse the bundle will be pulled around the bend point you just created. *Fig 58.6* shows 2 bundle bends creating an S between their points.



*Fig 58.6 Bundle bend nodes*

Alternatively you can make use of the **Bend Bundle** command  which will rotate the harness -

click on the **Bend Bundle End**  command followed by clicking on a section of bundle followed by the node on the end you would like to rotate then move the mouse in the direction and angle you wish to rotate the harness. *Fig 58.7* the bundle has been bent once at 270° setting it vertical then again at 0° setting it horizontal. *Fig 58.8* shows the Harness bent to fit within our A0 Frame with the bend points clearly marked in red.

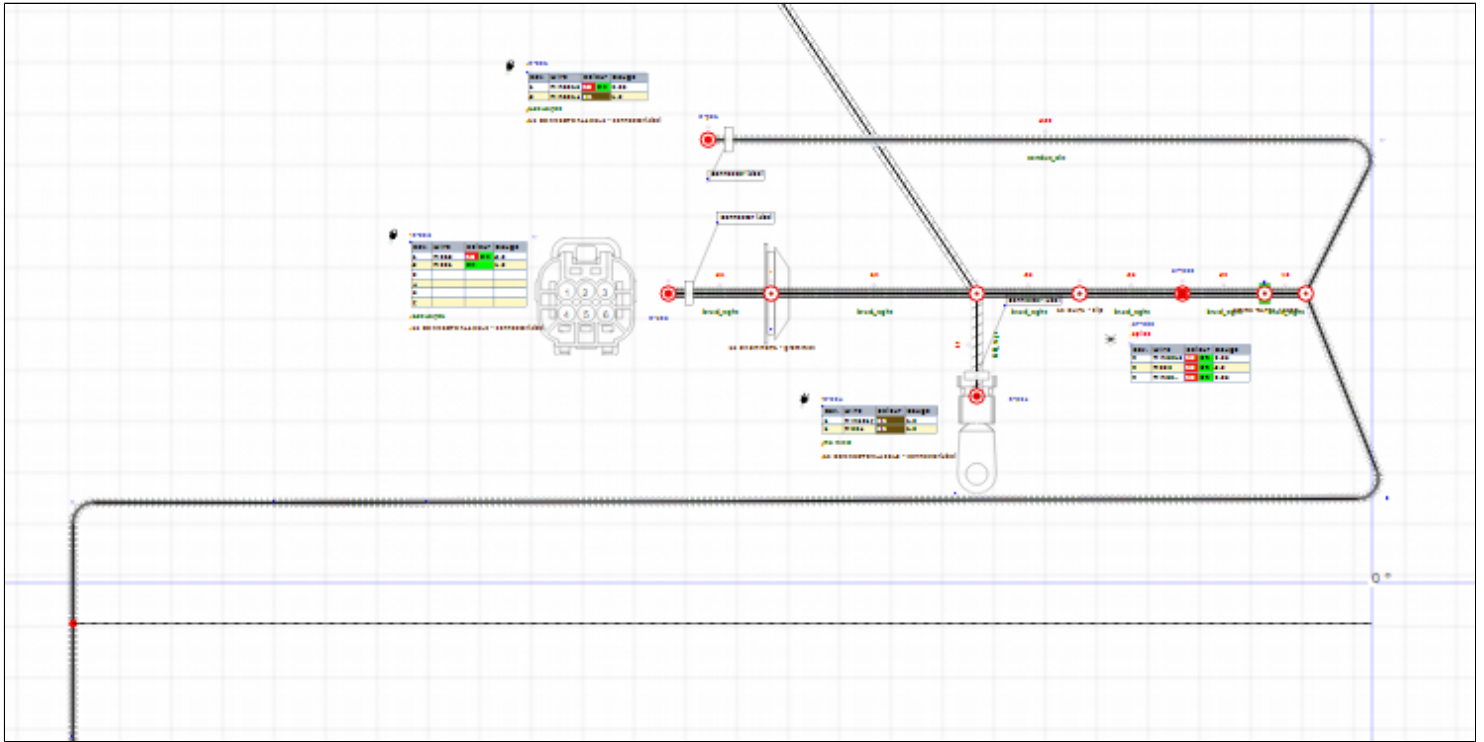


Fig 58.7 Bend bundle 270° and 0°

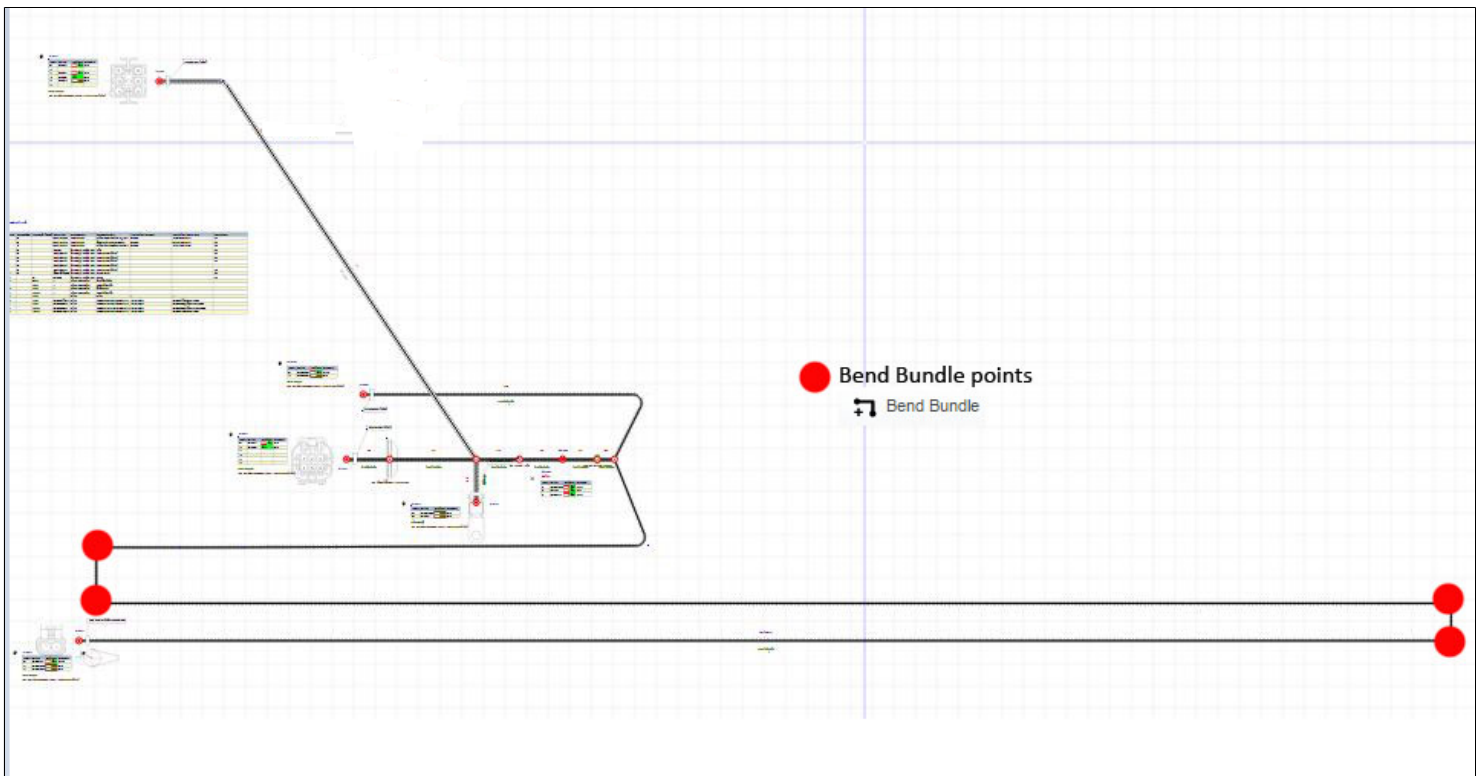


Fig 58.8 Bundle bent to fit A0 sheet.

## 59. Creating Tooling Components

To Create tool components which can be added to your bill of materials (Tooling). Go to Project Manager page (**Fig 59.1**), Select **Library**. Create a new Library (**TOOLING** in **Fig 59.2**) or add to an existing library by selecting your existing **Library Name** from the projects list OR clicking to create a new library for Harnesses Tooling parts (**Fig 59.2**)

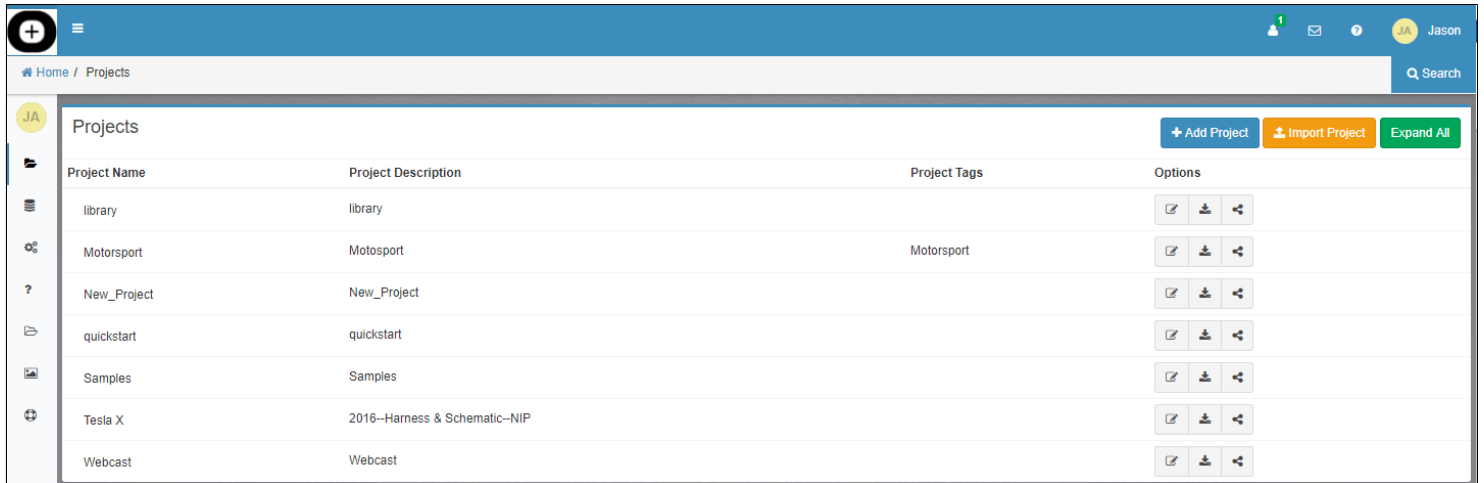


Fig 59.1 Projects Manager page.

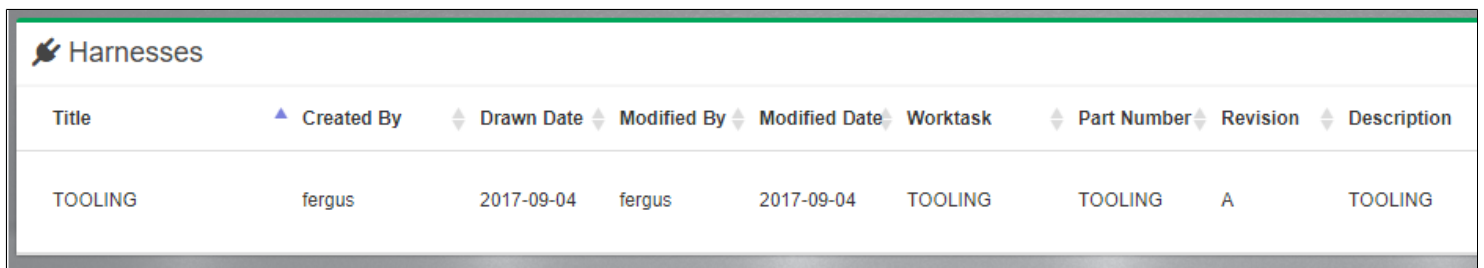


Fig 59.2 Library Details page showing Harnesses Tooling Library.

Add a new Sheet  then use the drawing tools to create a marker, the marker shape and size will be dictated by the shape of your component tool.

We will first create a nail using the **Drawing** Tab  and selecting the **Circle** tool then clicking and expanding to the desired size in this case is 18mm in size (**Fig 59.3**)

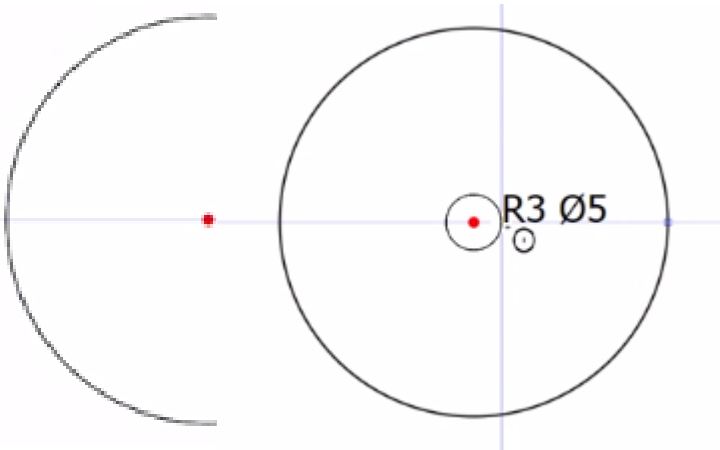


Fig 59.3  
Expanded  
circle 18mm.

Fig 59.4  
3mm Nail.

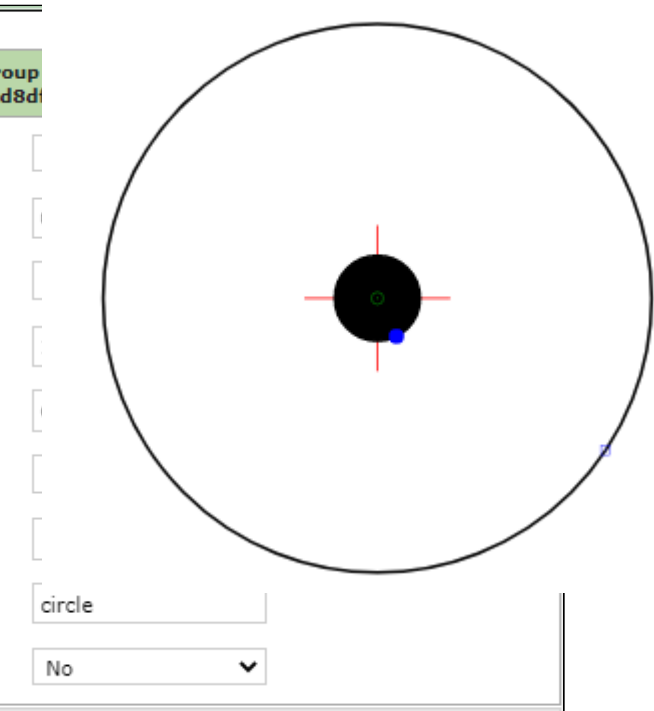
Again using the **Drawing** Tab draw a circle in the center for the Nail 3mm (**Fig 59.4**) it can be useful to add center lines to the circle to help with alignment, Select the circle and double click its control node, expand the **Advanced Parent options** and select Center Lines to **On** (**Fig 59.5**)

The image shows a software interface for editing a circle. The window title is "CIRCLE". It contains several sections of settings:

- Parent Options (Group 64 Element 342830b6b11a11ed8df602de8e05bcf8)**: This section is highlighted in green and contains:
  - Stroke: BLACK (dropdown)
  - Stroke width: 0.2 (text input)
  - Line Style: Solid (dropdown)
  - Opacity: 1 (text input)
  - Part Name: CIRCLE1 (text input)
  - Component DB: Manual (dropdown)
  - Part Number: (empty text input)
  - PN Description: circle (text input)
  - Show on BOM: No (dropdown)
- Variants**: (collapsed)
- Options**: (collapsed)
- Advanced Parent Options**: This section is highlighted in green and contains:
  - Centre Lines: Off (dropdown)
  - Layer Index: 50 (text input)
  - Group: 64 (text input)


*Fig 59.5 Circle drawing properties.*

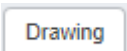

Next we will add details and colours to the Component tools for easy reference when added to our Harness drawing. To fill the nail, select its control node and double click to open the properties (**Fig 59.6**), set the Fill to **BLACK** and Submit to complete. (**Fig 59.7**)

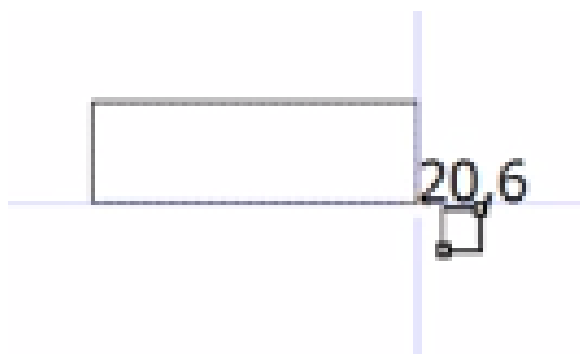


*Fig 59.6  
Nail circle  
Properties.*

*Fig 59.7 Filled  
Nail with  
center lines.*

Next we will create a Clip tool, Add a new Sheet by clicking the **Left Menu**  then the **"Add"** to add a new sheet.

Using the **Drawing** Tab  and selecting the **Rectangle** tool  Click and expand a rectangle 20mm x 6mm (**Fig 59.8**) and for the Center of the Clip draw a 3mm circle (**Fig 59.9**)



*Fig 59.8. 20 x 6mm Rectangle.*



*Fig 59.9. 20 x 6mm Rectangle with center circle and center lines.*

Adding outer limits to components improve its visibility when placed on the Drawing, again using the **Rectangle**

 tool we will add Black bars to each end of our Clip (**Fig 59.10**)

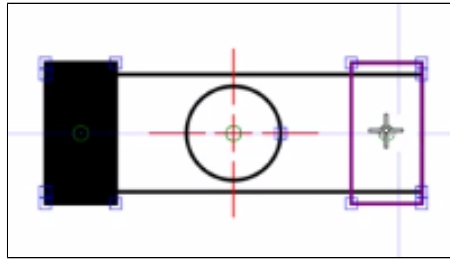




Fig 59.10 Blacked out limit bars on clip tool

Adding an image to a component tool can help visualise the part being built onto the drawing, to add an image under the **Drawing** tab click Image  the **Add link** properties will open, setting the frame to OFF will give a cleaner image when displayed on the drawing, select  **Image address:** browse to the location of the image either on your system or under the images stored within Arcadia.

We will add the Clip.jpg image under the Quickstart\tools folders (**Fig 59.11**)

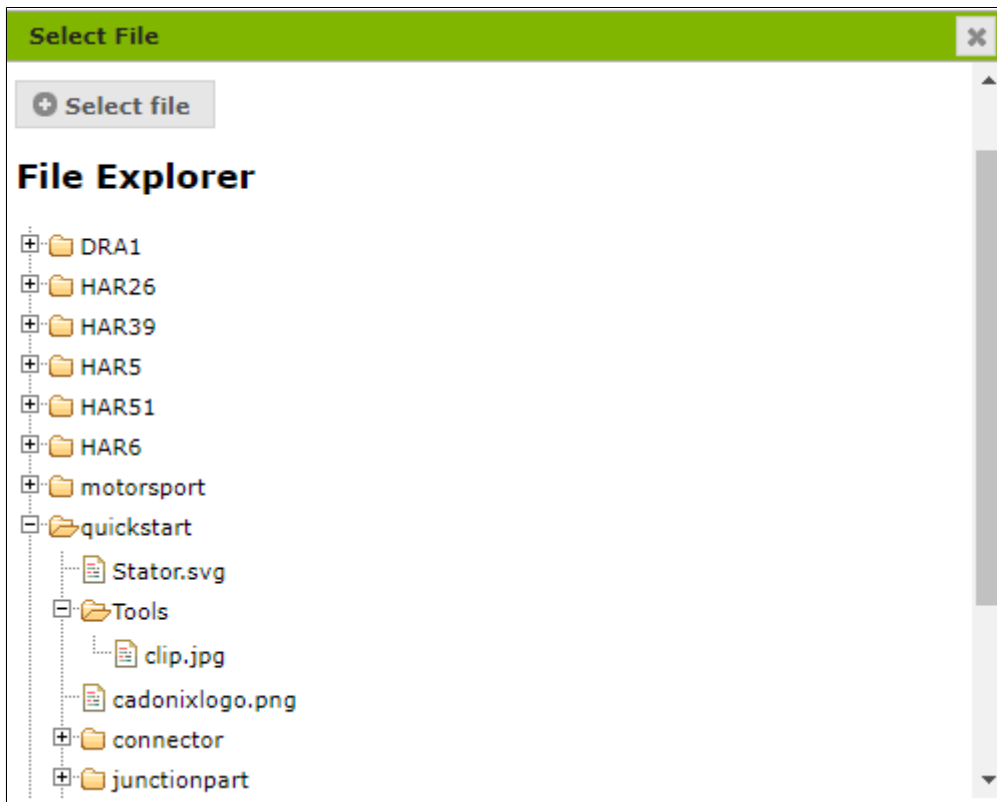


Fig 59.11 Image location properties

Place the image on the Sheet near to the component tool, the image may be too large in proportion to your tool so can be resized by double clicking the Images control node under **Advanced Parent Options** set the **Image scale** to your desired size (**Fig 59.12**)

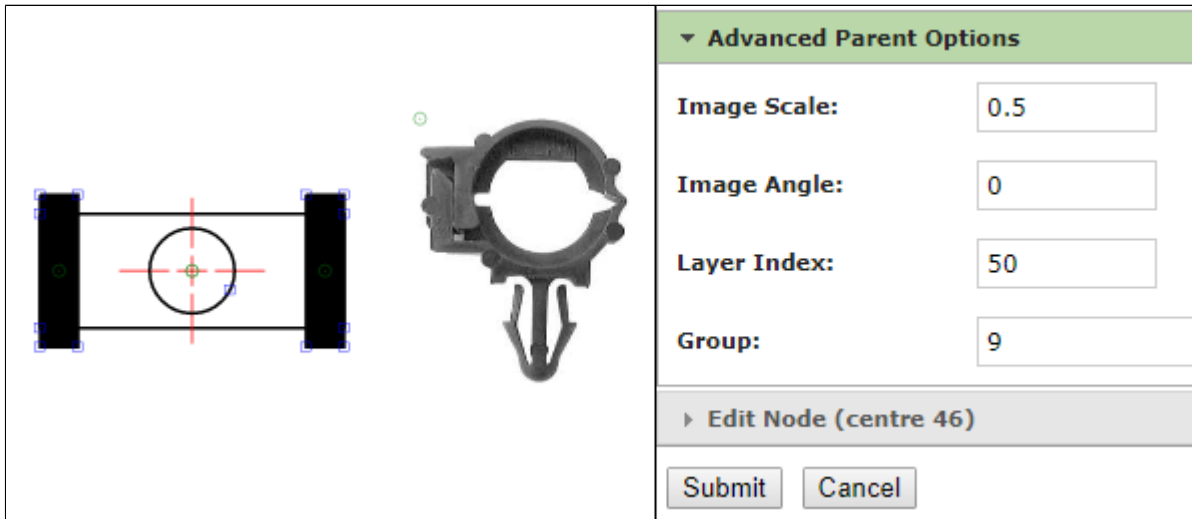



Fig 59.12 Image Advanced properties

Grouping drawn component tools merges them together and allows you to designate a center point. We will group our drawn clip tool along with its image Tool by Selecting using **Select**  **Select** and pulling a box around both then right clicking within the selection area, click Group (**Fig 59.13**) Set the center point to the center of your component tool and Submit to complete.

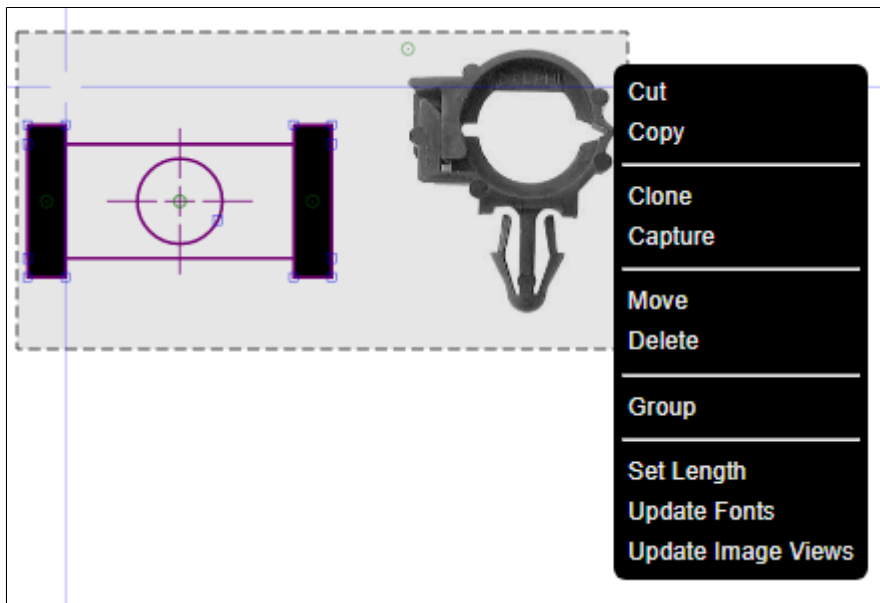
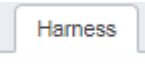
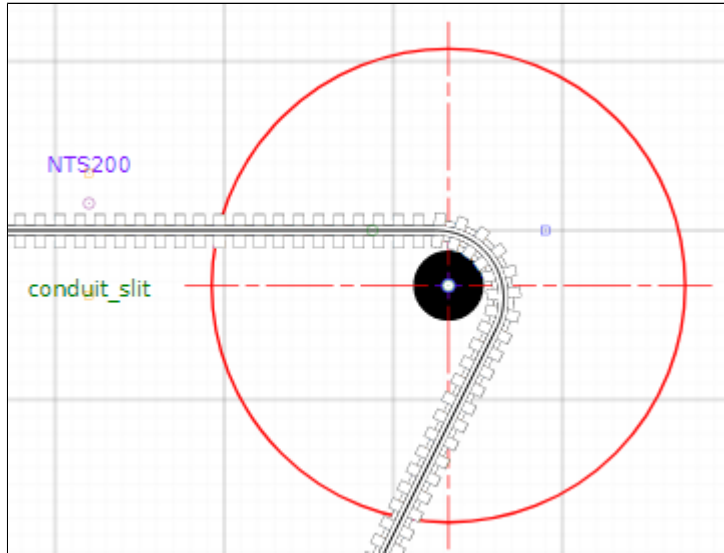
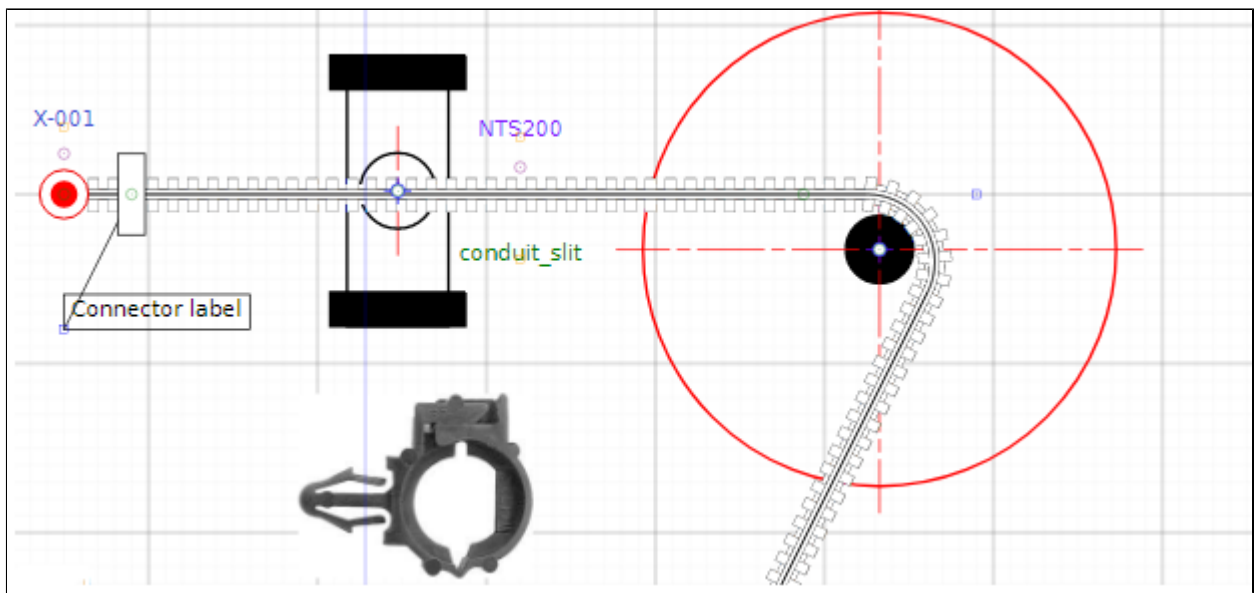


Fig 59.13 Selected area right click Menu

We will place our clip and nail onto our Fog Harness Formboard by reopening our Fog Harness formboard project, (Click the + / - next to **Fog Lamp Harness** and selecting the drawing below. Once opened Click the **Harness** tab  and then the **Library** icon, we can then select our nail and place it in the bend of conduit NTS200 as shown in **Fig 59.14** and complete the same process this time selecting the clip and placing it slightly further along Conduit NTS200 (Right click to rotate the clip) shown in **Fig 59.15**.



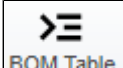
*Fig 59.14 Nail placed on Formboard*



*Fig 59.15 Clip rotated and placed on Formboard*

## 60. Formboard Tooling BOM

To Add a bill of Materials table to your Formboard containing or solely showing your Component tools, Click the **Options**



tab and **BOM Table** [BOM Table](#), Place the table on the Formboard then double click the tables control node to open the **BOM Table** (Fig 60.1) by changing the table type to Formboard or Both the list will contain all Component tools and their locations on the Harness (Fig 60.2)

**BOM TABLE**

▼ Parent Options (Group 62 Element cb0f9641b12111ed8df602de8e05bcf8)

Part Name:	<input type="text" value="BOMTABLE1"/>	Component DB:	<input type="text" value="Manual"/>
Part Number:	<input type="text"/>	PN Description:	<input type="text" value="bomtable"/>
Show on BOM:	<input type="text" value="No"/>	Variant Type:	<input type="text" value="ALL"/>
Part Category:	<input type="text" value="None"/>	Table Type:	<input type="text" value="HARNESS BOM"/>
BOM Table Header:	<input type="text" value="Top down"/>	Show Multiple PN:	<input type="text" value="HARNESS BOM"/>
Show Multiple	<input type="text" value="No"/>	Group by Connector / Splice:	<input type="text" value="No"/>
Company:		Table Group by:	<input type="text" value="Part Number"/>
Table Sort by:	<input type="text" value="Comp type"/>		

▶ Table Layout

▶ Table Properties

▶ Advanced Parent Options

▶ Edit Node

Fig 60.1. Formboard BOM options.

FORMBOARD BOM								
S.No	Quantity	Length (mm)	Part No.	Part Type	Description	Supplier Name	Supplier Part No.	Location
1	1		CLP2345	ANCHOR	Clip			[ZONE1/5B]
2	1		NL1234	CIRCLE	NAIL			[ZONE1/6B]

Fig 60.2. Formboard BOM Tooling Table.

## 61. Quick Add Functions

Now you will navigate back into the Quick Start Complete Harness to draw a new harness using the **Quick Add** functions, you will input the Bundle Layout, Covering, Junction Piece & Connector that includes a Backshell.

The connector that will be used in this example has been set up for your use within the Quickstart Library and already has the Base-Point's set for the Connector and Backshell, Within this example you will see how quickly it is to create a harness using the Quick to Add functionality.

Draw a T shaped Harness below the **Quickstart Harness Frame** with the dimensions shown on the image below (**Fig 61.1**)

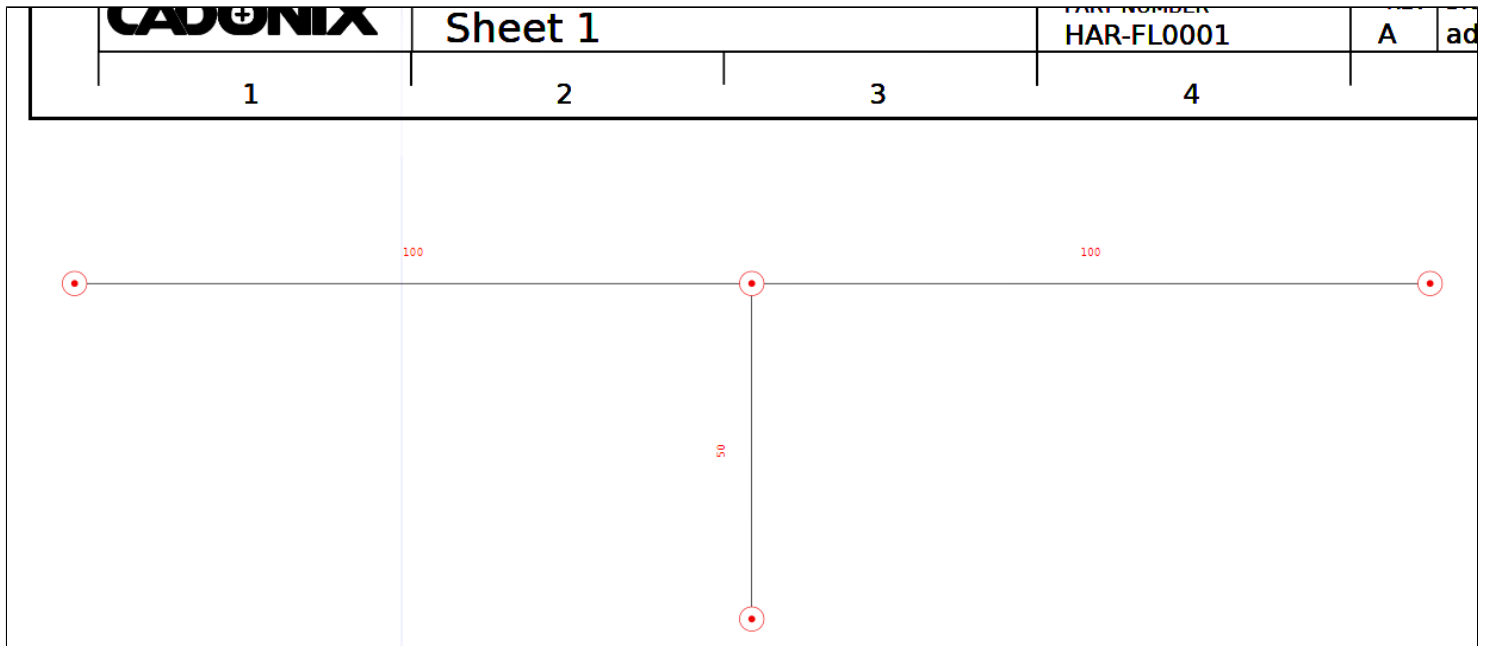


Fig 61.1 Draw the Harness Layout to lengths shown

Now you will Right Click the bundle and use the Add Covering **Add Covering** function and assign the part numbers to the Bundles (**See Fig 54.2.1, 54.2.2**):

**Part Number: 1967753**

**Description: UNSLIT CONDUIT NW 13/UFW PP-BS**

**Part Number: 1967753**

**Description: UNSLIT CONDUIT NW 13/UFW PP-BS**

**Part Number: 1967750**

**Description: UNSLIT CONDUIT NW 10/UFW PP-BS**

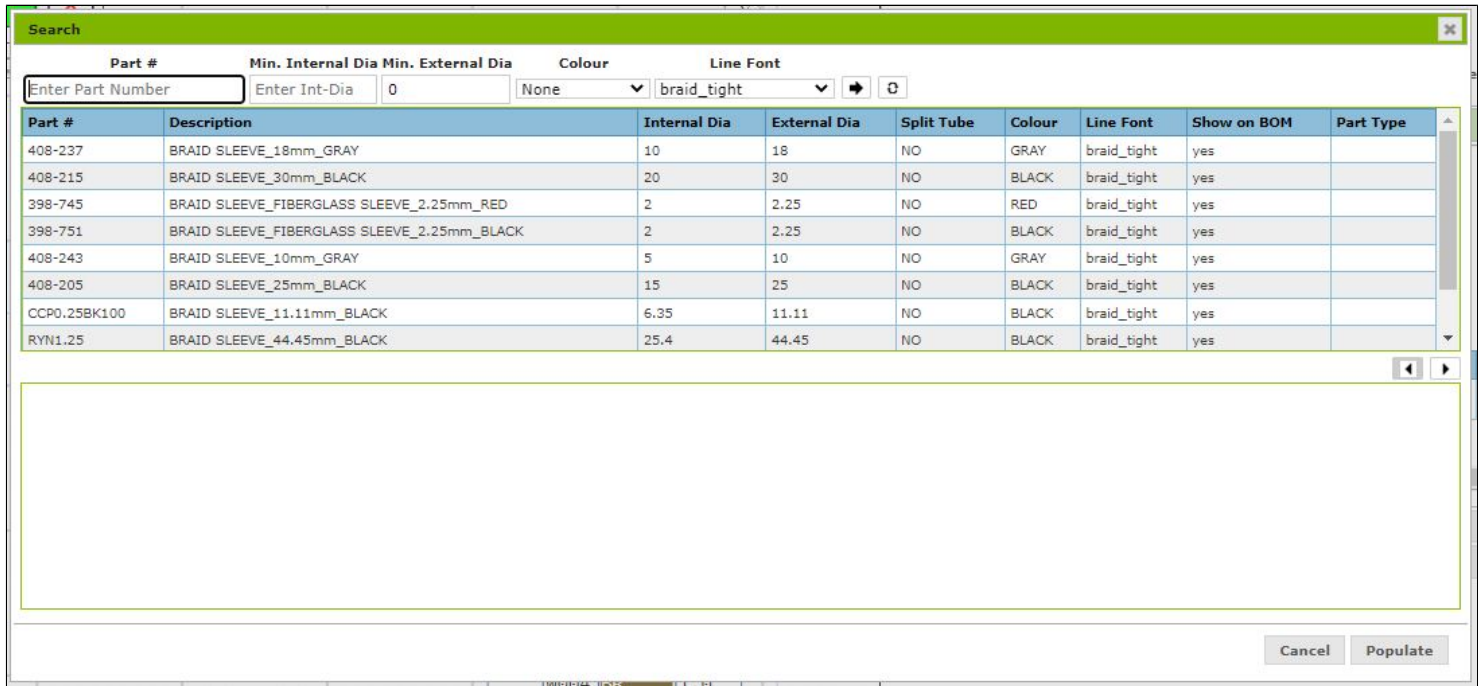


Fig 61.2.1: Right click to Add Covering will take you to Coverings library.

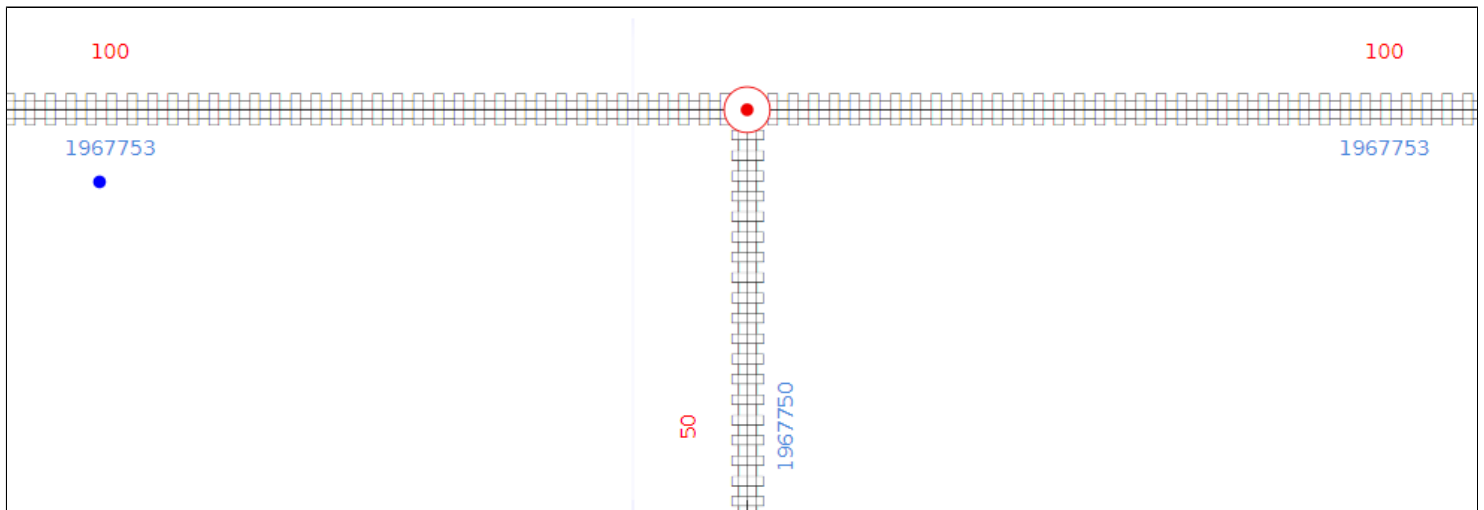


Fig 61.2.2: Please note the part numbers used for each branch of our Harness Layout.

Now you have assigned the Bundles with the Covering (13-10-13) you will now assign a Junction Piece from the Quickstart Library:

**Part Number: 9806101**

**Description: TPEICE 13-10-13**

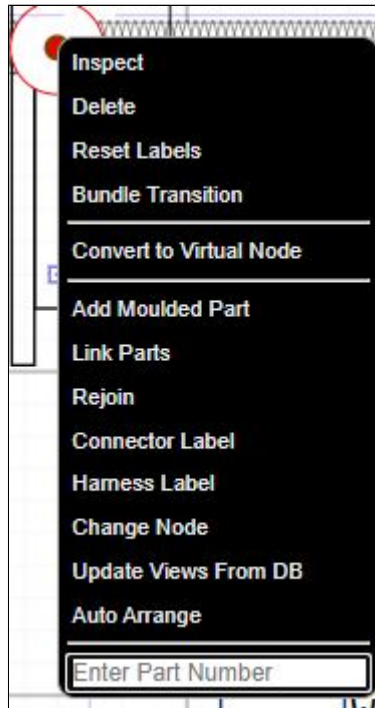


Fig 61.3: Right click to add any additional components to the Node.

Press enter, as this is the only component with this part number it will automatically assign the component to the node and place the Junction Part in the correct orientation of the bundle.

Now you will assign a Connector and enable to display the Backshell within the Attached Parts.

**Part Number: 12345605**

**Description: PLUG\_31WY\_HDP20\_BK\_SLD\_FEM TERMS**

**Attached Part Number: HD30-24BT-BK**

**Attached Description: BOOT ST\_HDP20-HD30\_PLUG\_RECIP\_SHELL SIZE 24\_BLACK**

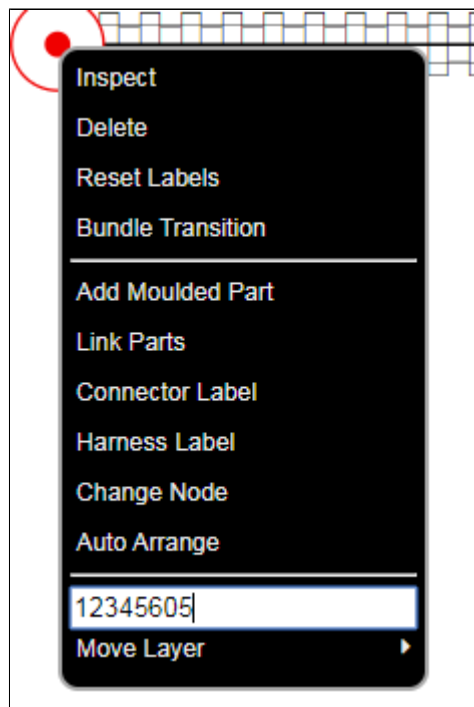


Fig 61.4: Press enter to search the component, If the part number does not exist you will be prompted, If there are multiple then they will appear within the search, If there is only One example this will be automatically selected to reduce any additional clicks.

Attached Parts														
Option	Component DB	Part Number	Part Type	Sub Part Type	Description	Qty/Length (mm)	Measure	Show on BOM	Requirement	Image Path	Default View	Show Image	Image Scale	Image
<input checked="" type="checkbox"/>	quickstar	12065279			TPA_METRI-PACK_2WY_BLACK	1	EACH	yes	Mandatory	noImage.svg	Top	no	1	0
<input checked="" type="checkbox"/>	quickstar	15-117-103			COVER- WIRING CONNECTOR	1	EACH	yes	Manual	noImage.svg	Top	no	1	0
<input checked="" type="checkbox"/>	quickstar					1	EACH	yes	Manual		Loading	no	1	0

Add Part No. Load Data from Database

Fig 61.5 : Tick the option to include the attached part & then press Submit.

BACK

X-006

100

1967753

Cav.	Wire	Colour	Gauge	Cav.	Wire	Colour	Gauge
1				26			
2				27			
3				28			
4				29			
5				30			
6				31			

Fig: 61.6 Tip: Use CTRL before clicking the Connector Image to move the Image by the Centre Point.

You have now completed the Quick to Add Function Guide.

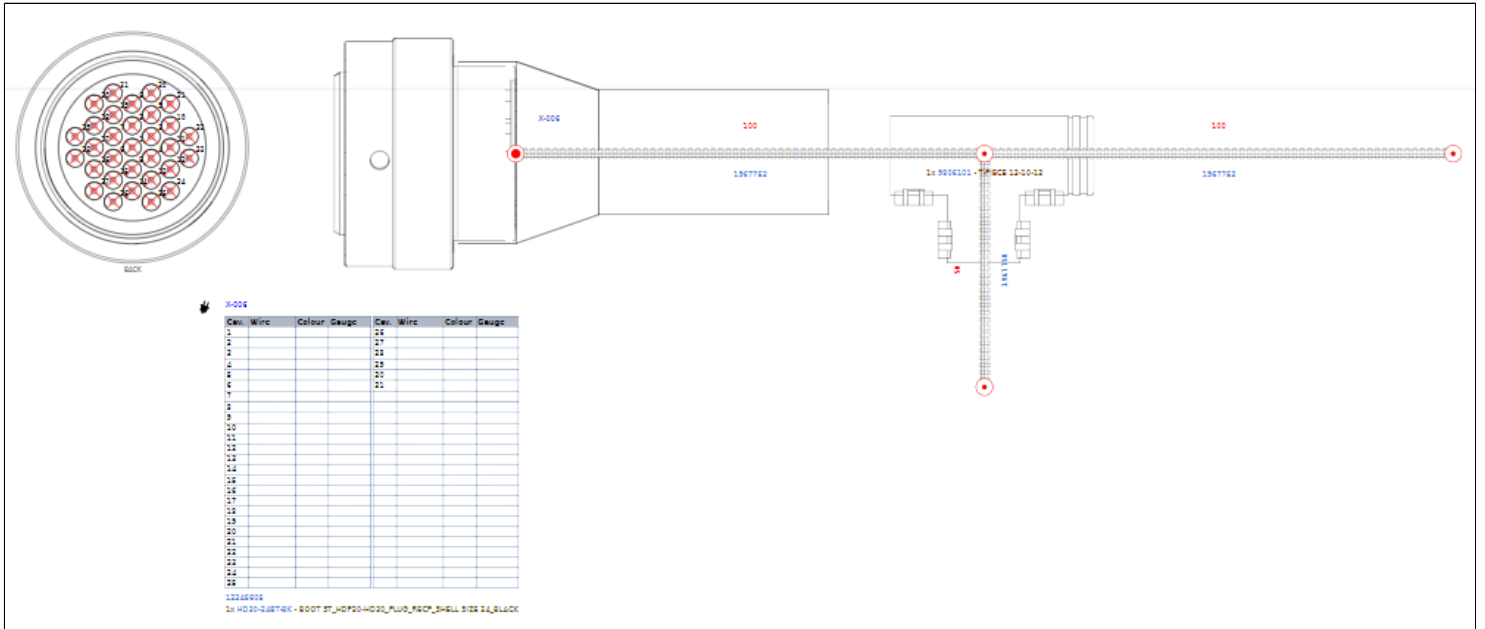
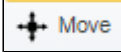



Fig 61.7 : If you wish to adjust the scale of these items you can do so by Right Clicking the Image and adjusting the scale by a numerical scale, Or you can select the move  function and scale the item by hovering over the image and selecting the adjust scale icon  and scaling with your Mouse movement.

---

## Quick Start Guide Conclusion

You have now successfully completed this Quick Start Guide. You have seen firsthand the capabilities and ease of use Arcadia provides. Now that you have scratched the surface please continue to use the tool and discover its true productivity.

### Need a Little More Help?

Should you have any questions please contact your local distributor or a member of our technical team and we will do our very best to help.

For all other enquiries please contact the Cadonix UK Headquarters.

Cadonix Ltd John Eccles House, Robert Robinson Avenue, Oxford Science Park, Oxford, OX4 4GP

Telephone: +44 (0) 845 591 0038 | Email: [enquiries@cadonix.com](mailto:enquiries@cadonix.com).