



Pulsonix Design System V10.0 Update Notes

2 Pulsonix Version 10.0 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2018

Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

Copyright in the whole and every part of this software and manual belongs to WestDev Ltd. and may not be used, sold, transferred, copied or reproduced in whole or in part in any manner or in any media to any person, without the prior written consent of WestDev Ltd. If you use this manual you do so at your own risk and on the understanding that neither WestDev Ltd. nor associated companies shall be liable for any loss or damage of any kind.

WestDev Ltd. does not warrant that the software package will function properly in every hardware software environment.

Although WestDev Ltd. has tested the software and reviewed the documentation, WestDev Ltd. makes no warranty or representation, either express or implied, with respect to this software or documentation, their quality, performance, merchantability, or fitness for a particular purpose. This software and documentation are licensed 'as is', and you the licensee, by making use thereof, are assuming the entire risk as to their quality and performance.

In no event will WestDev Ltd. be liable for direct, indirect, special, incidental, or consequential damage arising out of the use or inability to use the software or documentation, even if advised of the possibility of such damages.

WestDev Ltd. reserves the right to alter, modify, correct and upgrade our software programs and publications without notice and without incurring liability.

Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 10/08/18 iss 4

Pulsonix

20 Miller Court
Severn Drive
Tewkesbury Business Park
Tewkesbury
Glos, GL20 8DN
United Kingdom

Support Phone +44 (0)1684 296 570

Sales Phone +44 (0)1684 296 551

Fax +44 (0)1684 296 515

Email info@pulsonix.com

Web www.pulsonix.com

Contents

CONTENTS	3
VERSION 10.0 UPDATE SUPPLEMENT	6
Installing the New Version of Pulsonix	6
Licensing	6
Version 10.0 Update Notes	6
NEW IN VERSION 9.1 - NOT PREVIOUSLY DOCUMENTED	7
Page Link Attribute on Blocks and Block Ports.....	7
Save configuration Files - Speed Increase When Saving Over Network	7
Intelligent Gerber Import option - Add unmatched drill holes as pads	8
Component Search Engine.....	8
Product Version/Build Requirements	8
Registering with Component Search Engine	8
Downloading the Library Loader.....	9
Using the Pulsonix Component Search Engine	11
Updating Imported Parts to Company Standards.....	12
Component Search Engine and Pulsonix in Dual Screen Mode	12
Closing the Library Loader Application	13
NEW IN VERSION 10.0 - NOT PREVIOUSLY DOCUMENTED	14
Accel Netlist Import – Part Mapping Dialog	14
Netlist Output to SiMetrix Spice Simulator	14
Current Selection Mask Shown on the Status Bar	15
STEP Output – Settings Menu Change	16
STEP Output – Define Silkscreen Layer.....	16
Fillet and Mitre Corners for Differential Pairs and Bus Route.....	16
Differential Pairs - Fillets	16
Bus Route – Fillets	18
ODB++ Output includes Profile Cutouts	19
CAM Plots – Auto Generate Plot Ordered By Variants	20
Report Maker	20
Rotation of Attribute Command	20
Indent & Unindent Commands	21
NEW IN VERSION 10.0	23
Technology Dialog Changes	23
Enhanced Technology Dialog.....	23
Technology Dialog Tree Structure.....	24
New Style Naming pages for the Technology Dialog.....	24
Spacing Check of SMD Pads & Micro-vias	27
Pad Exceptions By Layer Class & Inner Electrical layers	28
Acid Trap Rules	28
Enclosure Spacing Rules	29
Select Net Names by Attribute	29
Variant Manager now a grid Dialog	30
New Adjacent Nets Rule	31
Define Pad Shapes now on Pad Styles dialog.....	32
Design Settings now in Technology Dialog.....	33
Grids now in Technology Dialog.....	33

4 Contents

Locked Grid Origins	34
Apply Origin To All Grids.....	34
3D STEP Preview Changes	35
Gradient Background (switchable between Solid and Gradient)	35
Improved Visualisation of the STEP Preview	36
Print option in STEP Preview	36
Toolbar with Icons of Pre-defined 3D Views	36
Perspective and Orthographic Views.....	37
Rotation of Design in STEP Preview now on Right Hand Mouse button.....	37
Selection of items in STEP Preview on Left Hand Mouse button	37
Move Component on Grid in STEP Preview	38
Automatically Synchronise 2D PCB from 3D	39
Working in both 2D and 3D Environments	39
Enclosure Spacing Rules	40
Adding Mechanical Enclosures	40
Transparency Setting for Enclosures	41
Colour Setting for Enclosures.....	42
Clash Detection within the 3D Environment	42
Using the Clash Detection Option	43
Colouring for non-coloured STEP Models	43
Properties of Selected design items shown on Status Bar.....	44
Speed Increase of STEP file Generation through Multithreading.....	44
STEP Model to Footprint Alignment.....	44
3D Package Built-in Attribute.....	45
Selection Mask.....	45
Updated User Interface	47
4K Graphics Support	47
Simplified Default Toolbars	47
More Readability for Read Only Edit Boxes	47
Design Attributes in Variants.....	48
Interactive Copper Pour Healing.....	49
Changes to Manual Routing Modes.....	50
Segment Modes	50
Edit Mitre Mode	51
Design Rules Checks Dialog Changes.....	51
New DRC Checks - Acid Traps and Adjacent Nets.....	51
Clear Warnings Option.....	52
Automatically Open DRC Errors Bar Switch	52
Attribute Editor Changes	52
Footprint Name in Edit Attribute Dialog	52
Hide Inbuilt Columns	53
Reload All – Do Not Reload Doc Symbols.....	53
New Commands in Options Dialog	53
Middle Mouse Wheel/Button Usage Options	53
New Component Pads Selection.....	54
Create PCB Group for SCM Block.....	54
Join Open Shapes.....	55
Save Doc Symbols To Appropriate Library.....	56
Changes to Error Markers.....	57
Find will Find Areas with no name.....	57
Find Bar Changes to Select All Find Items.....	57
Dynamic Starting for PCB Bus Routing	57
New Template Option – Clear On Edit.....	58
Library Manager Changes.....	58

Resizable Library Manager.....	58
Part Pin Mismatch Report.....	58
STEP File Names in Part Details.....	59
Clear Filter.....	60
Custom Indexing for Footprints.....	60
Manual list Refresh in Library Manager.....	62
‘Follow Item’ on a Part option.....	62
Find Symbol.....	62
Find Footprint.....	63
Find Footprint in 3D Packages.....	64
Rename in Library Manager – Replace Symbol References in Parts.....	65
Swap Symbols Option.....	65
Create New Libraries directly from the Library Manager Folder tab.....	66
Part Description Using Substitute Attributes.....	67
CAM Plots – Drill Output available for Board Outlines & Cutouts.....	67
CAM Plot – Plot String Attribute.....	68
Export IPC-2581 - Omit BOM Data.....	69
Toggle True Width For Layers.....	69
Report Maker Changes.....	69
Access Other Tables in PDC Database.....	69
Adjacent Nets in Report Maker – List of Errors.....	70
High Speed Option Changes.....	71
Differential Pairs - Turn Backwards direction on Via Pattern.....	71
Improved Accuracy of Differential Pairs.....	71
Improved Differential Pair Routing – Fillet Mode.....	72
Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components.....	73
Finish Track On Differential Paired Track (Spur).....	74
New Serpentine Skew Modes on Differential Pairs.....	74
Move All Vias in Composite Span.....	75
Network License Server (NLS) Changes.....	75
Default Supplied Technology Files Changes.....	76
Unattended Installation.....	76

Version 10.0 Update Supplement

Installing the New Version of Pulsonix

It is always recommended that you back-up all libraries, designs, technology files, profile files and report files before installing the latest version. Other than for any technical reason, this is good working practice, although you should already have a backup of this data!

To install Pulsonix, double-click on the download executable and wait for a short time. You may have been supplied a CD, in which case insert it and wait for it to run. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 10.0 on top of your existing V9.0 installation. If upgrading from V9.0 or V9.1, you can install it alongside the older version if you prefer. In any case, you do not need to uninstall the old version first unless you wish to remove it from your hard drive.

Licensing

Version 10.0 requires a new license if a new user or upgrading from any older version of Pulsonix.

If you are using a version of Pulsonix earlier than Version 9.0 or 9.1, you will require a new license, this would have been supplied to you under the terms of your maintenance contract.

For existing users upgrading from a previous version, it is recommended that you save the new license and overwrite the existing one. When requested during installation, simply click the **No Change In Licensing** check box on the licensing page of the installation wizard. The **License Manager** can be used to add new licenses and make changes to network licensing after the installation has been completed.

Version 10.0 Update Notes

Update Notes for Version 10.0 are available in the Pulsonix installation folder under \documents. Alternatively, select the direct link within the product itself: **Help menu | Online Manuals> Pulsonix V10.0 Updates Notes**

New In Version 9.1 - Not Previously Documented

The feature(s) below appeared in Version 9.1 but were not documented. They have been added here for your reference:

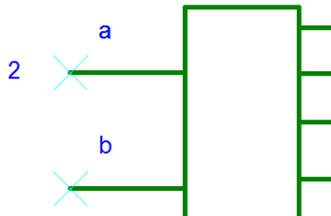
Page Link Attribute on Blocks and Block Ports

You can now use the page link attribute on **Blocks** and **Block Ports** to show the name of the page they link to.

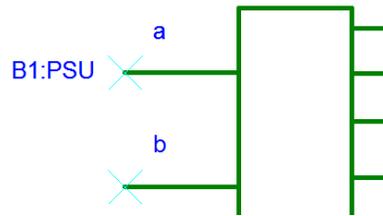
A new option on **Design Settings** and **Naming** page enables you to **show the Page Number** instead of the **Page Name** on a page link.

Prefix:	<input type="text"/>	<input checked="" type="checkbox"/> Display Connector Pin Name
Separator:	<input type="text" value="."/>	Connector Pin Separator: <input type="text"/>
Newline After:	<input type="text" value="0"/>	
<input type="checkbox"/> Show Current Page		
<input checked="" type="checkbox"/> Show Page Number (instead of Name)		
Page Link Attribute		Part Names
<input type="checkbox"/> Show Page Number (instead of Name)		<input type="checkbox"/> Display On Each Gate On Page

Page link on block showing Page Number



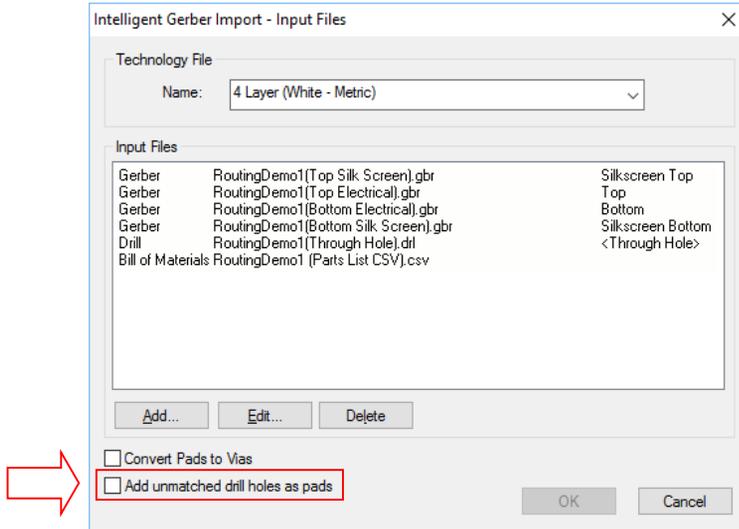
Page link on block showing Page Name



Save configuration Files - Speed Increase When Saving Over Network

A significant speed increase has been made for when saving your Configuration files to a network location. The speed is equivalent to that when saving the file locally now. This was added to a later version of Pulsonix 9.1

Intelligent Gerber Import option - Add unmatched drill holes as pads



With the **Add unmatched drill holes as pads** check box selected, the import option tries to match drill holes (in the drill file) with pads/flashes in the Gerber file. This enables data that has holes only but no pad, such as mounting holes, to be imported.

If the drill doesn't match a pad, it adds a pad for the drill hole even when it doesn't match a pad in the Gerber, otherwise a pad won't be added (and so no hole).

Component Search Engine

The **Component Search Engine** web site enables you to search Parts (along with Symbols and PCB Footprints) and download them to your Pulsonix application ready to use. Not only do they appear on the end of your cursor but are also automatically added to your library to use on another design.

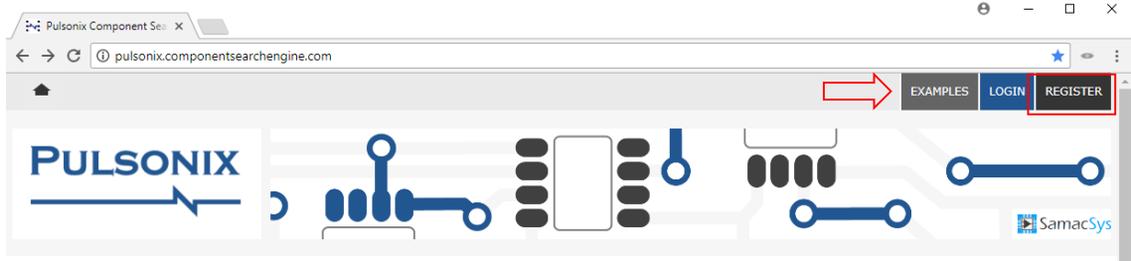
Product Version/Build Requirements

You will need to be running Pulsonix 9.1 Build 6871 onwards to benefit from the **Component Search Engine** integration. This feature also runs will all version of Pulsonix 10.0 and later.

Registering with Component Search Engine

To use the Component Search Engine, you will first need to register an account.

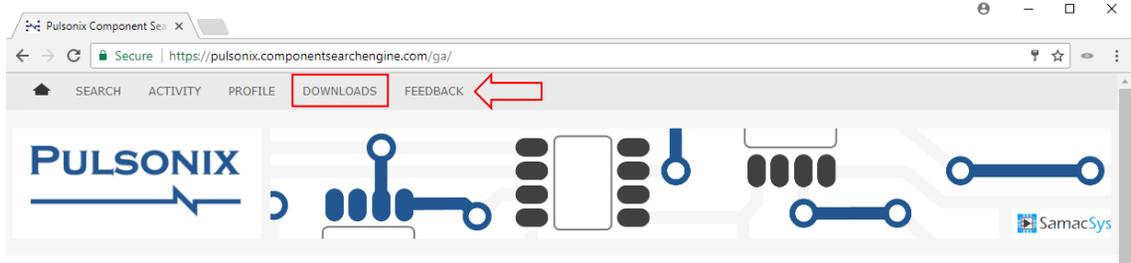
Go here - <http://pulsonix.componentsearchengine.com/> and click **Register**.



Downloading the Library Loader

Once you have registered, you are automatically logged in. You now need to download the **Library Loader** application. This is required as the interface between the Component Search Engine web site and the Pulsonix application.

Click on the **Downloads** button at the top of the Component Search Engine web page.



On the **Downloads** page, scroll down to the **Library Loader** section.

Pulsonix Downloads

ECAD Part Wizard

This Excel based tool provides the following funtions.

- Find and download models from our library
- Fracture the models to your requierments
- Preview the symbols and footprints prior to export
- Export to any of our supported CAD tools
- Request new models be created

[Download EPW version 1.93](#)

Library Loader

This helper application streamlines the process of loading our models into your CAD system.

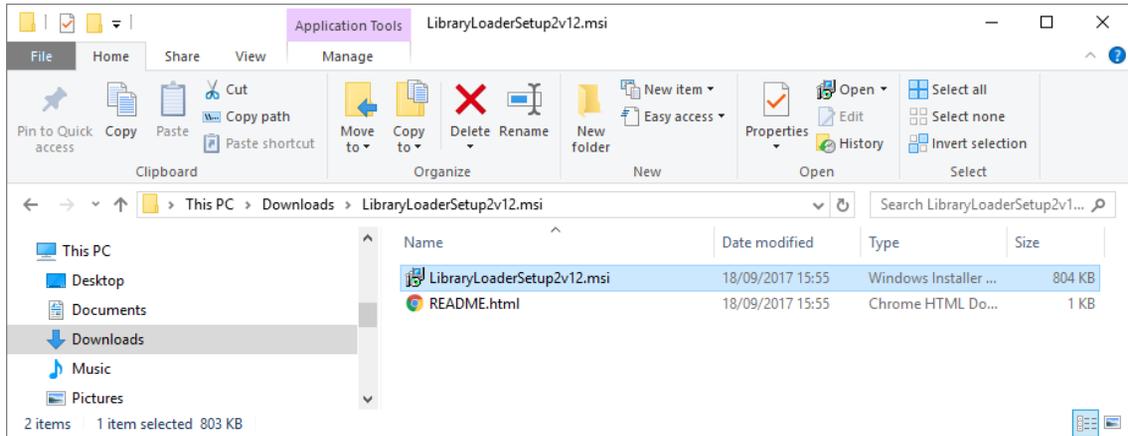
Simply click this icon  from any website and once the Library loader is installed the model will appear on your cursor ready to place on your design. From a website to your design in seconds!

[Download Library Loader version 2.12](#)

10 Pulsonix Version 10.0 Update Notes

Click the **Download Library Loader** button to start the download.

Once downloaded, run the **LibraryLoaderSetup.msi** program from your download folder.

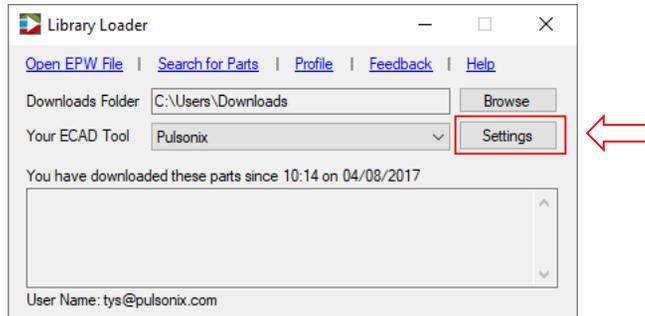


Once the **Library Loader** has been successfully installed, open the **Library Loader** from your Windows **Start** menu.

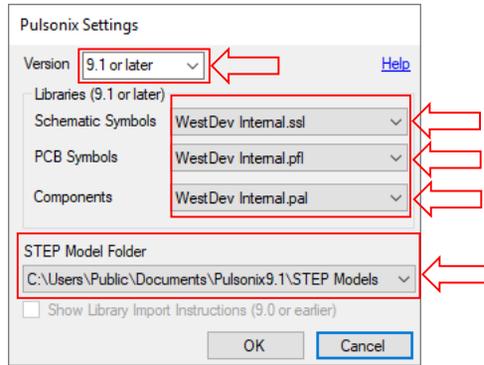
You will be asked to either Register or Login. As you have already registered you can select the **Login** Tab and **Login** using your **User Name** and **Password**.



Once logged in, you now need to click the **Settings** button on the **Library Loader** dialog, this will open the **Pulsonix Settings**.



From within the **Pulsonix Settings** dialog, you need to inform the Library Loader which version of Pulsonix you are using. If using 9.1 onwards, you will now be able to set the destination for where new Step Models, Schematic Symbols, PCB Footprints and Components (Parts) will be saved to.



Note: you need to be using Pulsonix 9.1 (Build: 6869) onwards to have Parts automatically saved from the Component Search Engine into your Pulsonix Libraries.

Now you've set up the Library Loader you can now download Parts from the Pulsonix Component Search Engine website and they will load directly into your libraries and onto the end of your cursor ready for placement in a design.

Using the Pulsonix Component Search Engine

From the **Pulsonix Component Search Engine** website (<http://pulsonix.componentsearchengine.com/>), enter the name of your Part or a description of it in the **Search** section. Once the search is complete you will see six columns presented including an ECAD Model, Datasheet and Pricing/Stock information for the Part from various suppliers.

Clicking the **Price/Stock** link will allow you to select and download the Part with pre-populated attributes from a specific supplier.

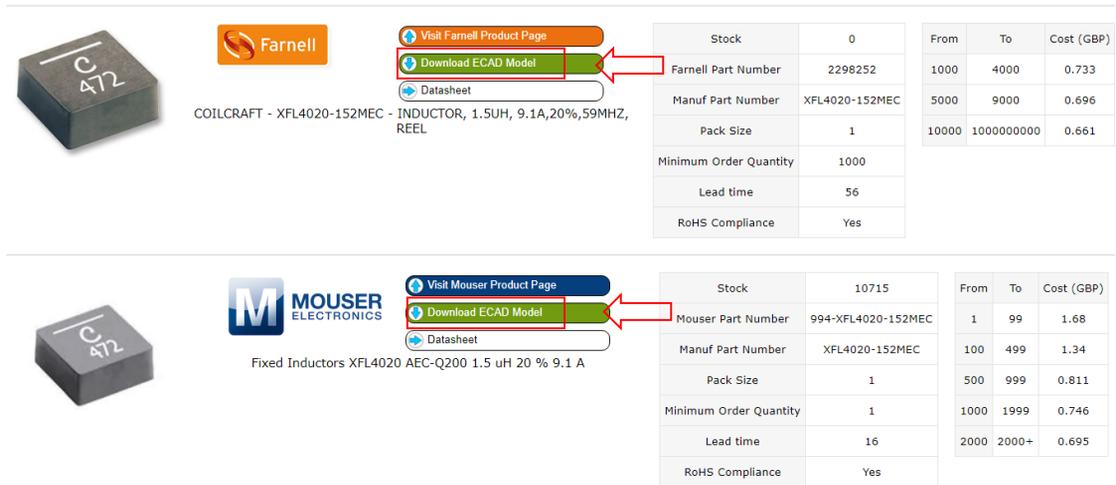
coilcraft

Components matching your search: 198506

Image	ECAD Model ↓	D.S.	Description	Manufacturer	Prices / Stock
	 C4		COILCRAFT - XFL4020-152MEC - INDUCTOR, PWR, 1.5UH, 9.1A,20%,59MHZ	COILCRAFT	XFL4020-152MEC

Clicking the **Download ECAD Model** button will download the model from the website into your libraries and onto the end of your cursor ready for placement in the design. Attributes are automatically added to the Part depending on the section you download it from, therefore if we click **Download ECAD model** from Farnell, we would then also see the Farnell Part Number on the part in Pulsonix.

12 Pulsonix Version 10.0 Update Notes



Farnell
COILCRAFT - XFL4020-152MEC - INDUCTOR, 1.5UH, 9.1A,20%,59MHZ, REEL

Stock	0
Farnell Part Number	2298252
Manuf Part Number	XFL4020-152MEC
Pack Size	1
Minimum Order Quantity	1000
Lead time	56
RoHS Compliance	Yes

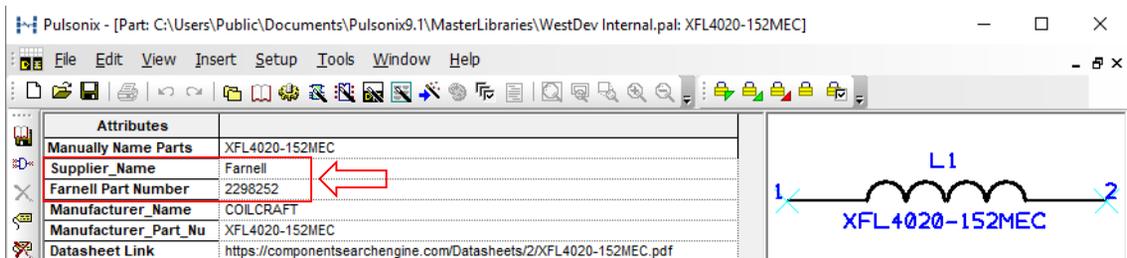
From	To	Cost (GBP)
1000	4000	0.733
5000	9000	0.696
10000	100000000	0.661

MOUSER ELECTRONICS
Fixed Inductors XFL4020 AEC-Q200 1.5 uH 20 % 9.1 A

Stock	10715
Mouser Part Number	994-XFL4020-152MEC
Manuf Part Number	XFL4020-152MEC
Pack Size	1
Minimum Order Quantity	1
Lead time	16
RoHS Compliance	Yes

From	To	Cost (GBP)
1	99	1.68
100	499	1.34
500	999	0.811
1000	1999	0.746
2000	2000+	0.695

Now the Part has been downloaded, you can investigate its attributes from the **Edit Part dialog** to see it has inherited the values from the online model, including the Supplier name and Supplier Part number.



Attributes	Value
Manually Name Parts	XFL4020-152MEC
Supplier Name	Farnell
Farnell Part Number	2298252
Manufacturer Name	COILCRAFT
Manufacturer Part Nu	XFL4020-152MEC
Datasheet Link	https://componentsearchengine.com/Datasheets/2/XFL4020-152MEC.pdf

L1
XFL4020-152MEC

Updating Imported Parts to Company Standards

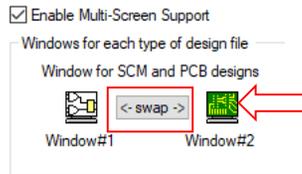
It is always worth remembering that imported parts may need to be updated to your company standards. For example, this may mean adding attributes to Parts or changing text, pin and line styles on Symbols and Footprints to be in-line with your other library items.

This concludes the instructions for using the Pulsonix Component Search Engine.

Component Search Engine and Pulsonix in Dual Screen Mode

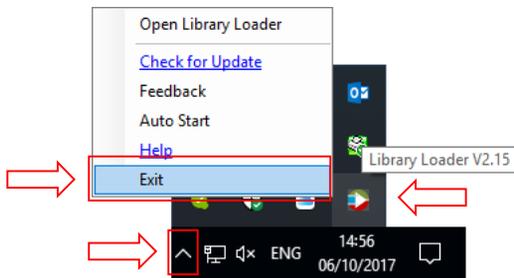
If you are using Pulsonix in Dual Screen mode, the Component Search Engine will automatically place the component symbol in the 'First' screen.

To change which environment you would like the Search Engine to place the symbol in, go to **Pulsonix, Options dialog, Multi-Screen** tab and click **Swap**.



Closing the Library Loader Application

As the Library Loader will run in the background, you will need to close the background application if you wish to close it completely. To do this, select the **up arrow** on your **taskbar**, select the **Library Loader** icon and from the menu displayed, click **Exit**.



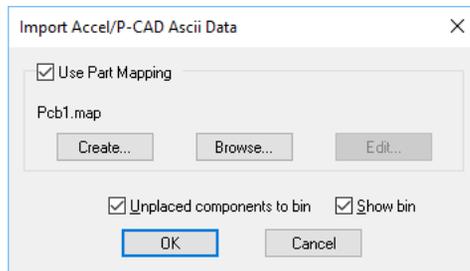
New In Version 10.0 - Not Previously Documented

The following features are new to Version 10.0 and have been added since the first release was created.

Accel Netlist Import – Part Mapping Dialog

When reading an **Accel Netlist** into Pulsonix PCB using the Accel Netlist Import facility from the **File** menu, the Part Mapping facility has been added. This enables the Part name to be mapped from one used in the netlist to one from your library.

Note: The import uses the *originalName* field in the *compInst* section in the Netlist for the part name.



On a first run of the dialog, if no mapping files already exist, press the **Create** button. This will read the Parts in the Netlist and ask you for a name to save the mapping file to. You can edit the Parts with the dialog displayed.

Orig. Part	New Part	Orig. Footprint	New Footprint
C Polarised	C Polarised		
22FLZ-SM2-TB345678	22FLZ-SM2-TB345678		
R	R		

The **Browse** and **Edit** buttons enable you to choose an existing mapping file or edit an existing mapping file.

Note, the mapping file is an ASCII forms based file and can be edited if you understand the format and are careful.

Netlist Output to SiMetrix Spice Simulator

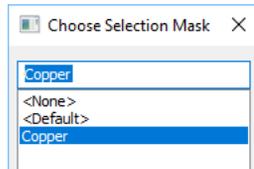
For users who have independently purchased SiMetrix Spice, there is now a specific output to write a netlist for the SiMetrix and SimPliS products. This is an extended version of the standard Spice netlist and is formatted for use with these simulators.

From the **Simulation** menu and **Set Netlist Spice Type**, use the **Spice Netlist Type** drop down option and select **SiMetrix**. If the **Run Simulator** check box is selected, it will populate the command line switches for you.

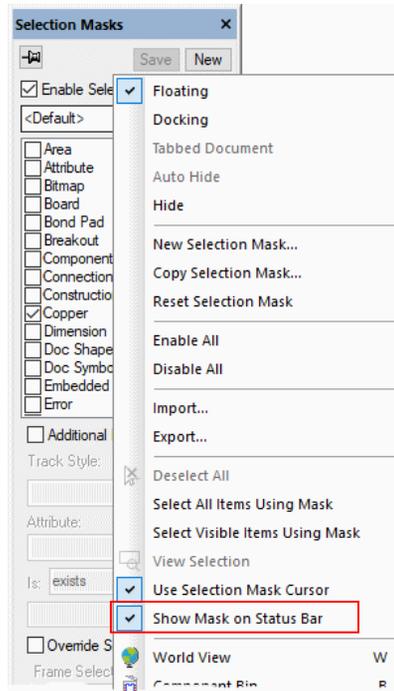
Current Selection Mask Shown on the Status Bar

The current **Selection Mask** being used is now shown on the **Status Bar**. Double click on it to choose a mask.

Once selected, it will display <None> if no mask being used (where the **Enable Selection Mask** switch is selected off in the main browser window), or <Default> if the default setting is being used or *User_Selection* which will be a named user-defined selection mask.

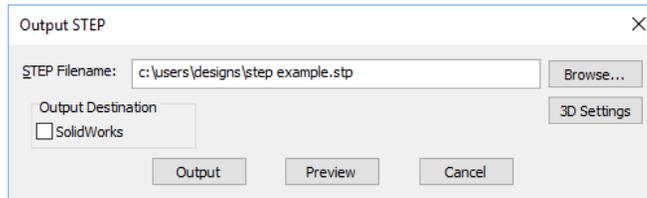


You can control whether it is displayed on the Status Bar by using the **Show Mask on Status Bar** switch on the main **Selection Mask bar**.



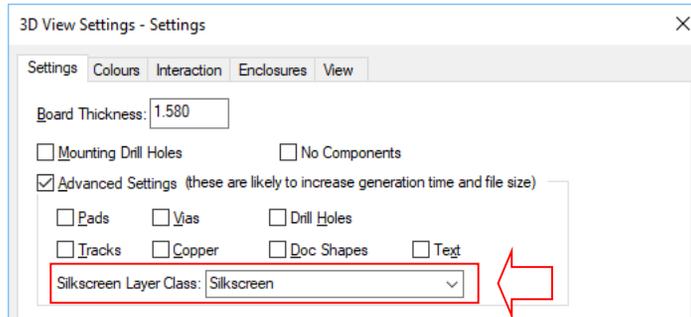
STEP Output – Settings Menu Change

In the release version of Pulsonix 10.0, there was a change to the **Output STEP** dialog which meant that the settings switches were moved from the main front dialog to an inner **Settings** tab of the **3D Settings** dialog. This hadn't been previously documented.



STEP Output – Define Silkscreen Layer

Previously, the STEP output was looking for the key word *Silkscreen* when outputting silkscreen data into the STEP file or STEP Preview window. This meant that if your silkscreen layer wasn't named 'silkscreen' or perhaps was been written in a different language, the silkscreen would not appear in the file. This has been replaced with the ability to define the **Silkscreen Layer Class** using the drop down list box displaying the **Layer Class** names in your **Technology**.



Fillet and Mitre Corners for Differential Pairs and Bus Route

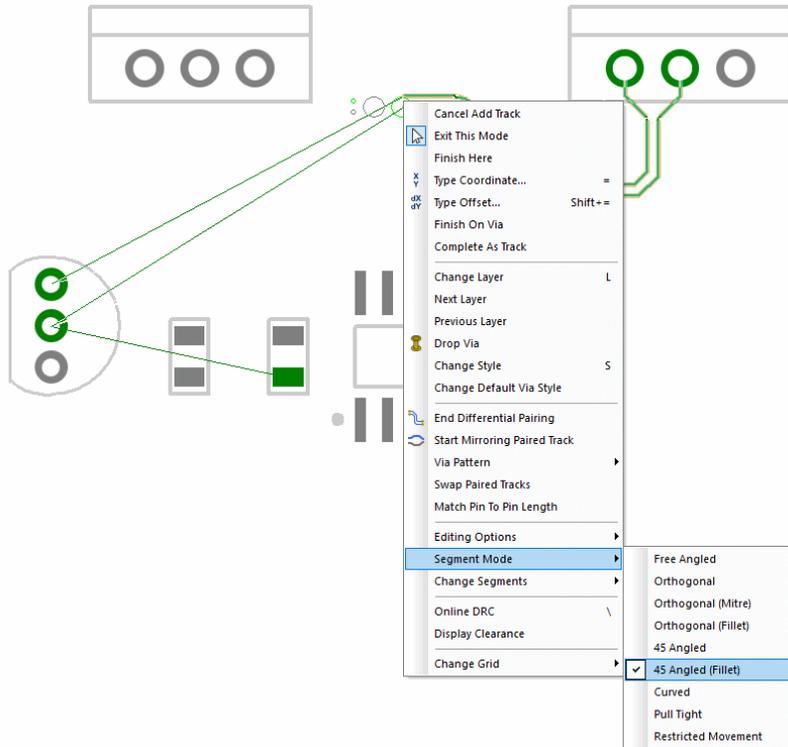
When using the Interactive High Speed option, you can now Fillet corners on a Differential Pair and Bus route.

Differential Pairs - Fillets

The ability to Fillet and Mitre corners in a Differential Pairs has been added.

Interactive Differential Pair Routing

During the routing of Differential Pairs, from the context menu, you can select the new Segment Mode of **45 Degree (Fillet)** to created filleted corners.

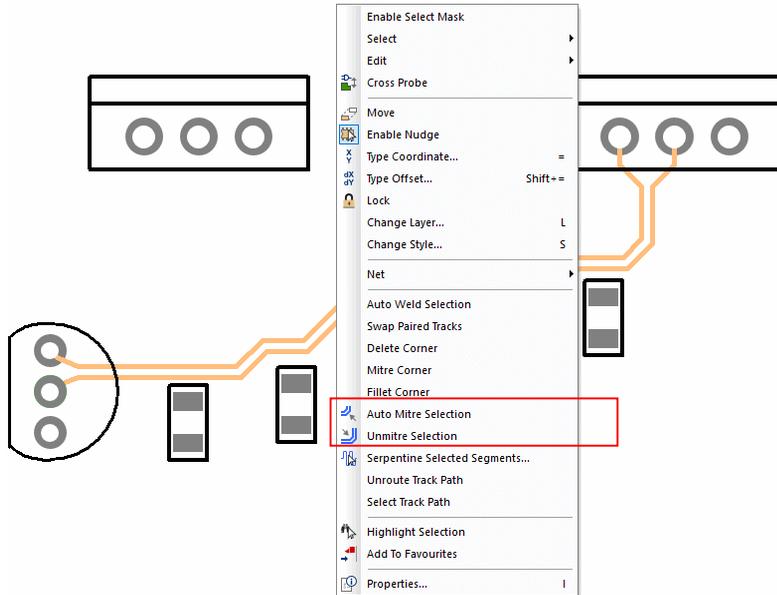


Existing Differential Pairs

With an existing Differential Pairs created, by selecting an existing mitred corner or the whole path (using Shift-select), you now have the ability to add Fillets to the corners.

From the context menu, two new commands exist, **Auto Mitre Selection** and **UnMitre Selection**. The **Auto Mitre Selection** option will use the settings from the **Technology** file, **Parameters**, **Auto Mitre** page.

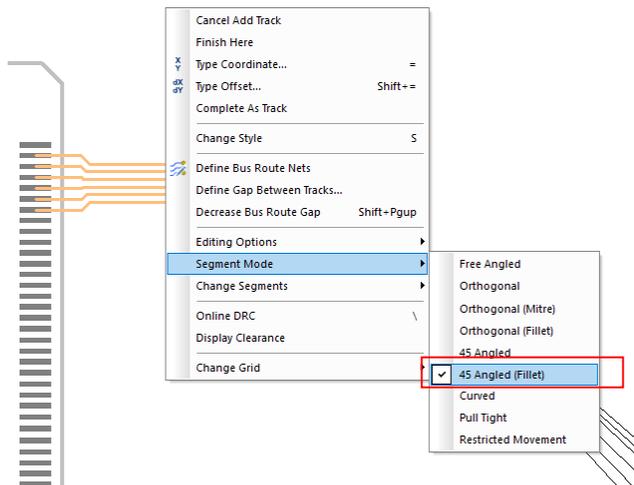
<input type="checkbox"/> Curved Mitres	Maximum Mitre Size: <input type="text" value="50.0"/>
<input type="checkbox"/> Any Angle	Minimum Mitre Size: <input type="text" value="12.5"/>



Bus Route – Fillets

Interactive Bus routing

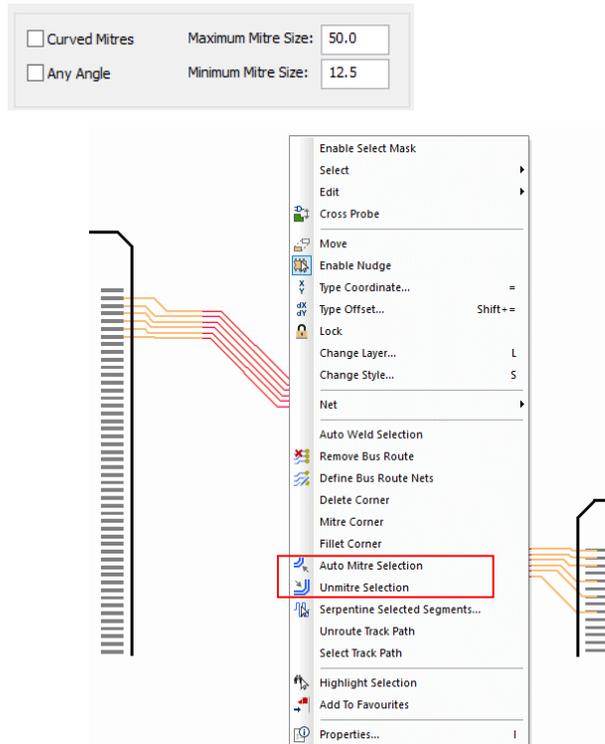
When interactively Bus routing in PCB, you can now switch to the new segment mode **45 Angled (Fillet)** using the option from the context menu.



Existing Bus Routes

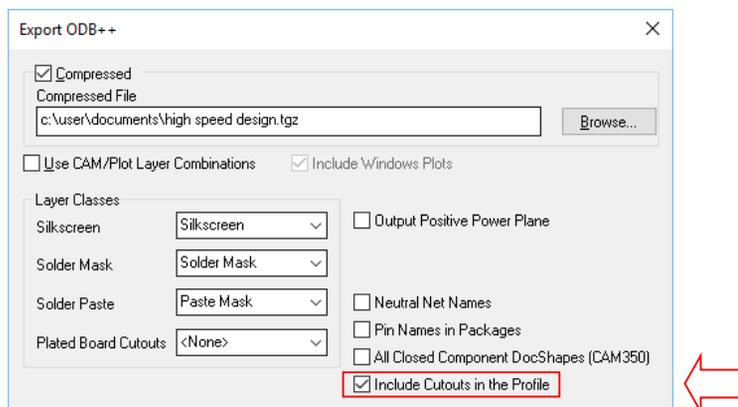
With an existing Bus route created, by selecting an existing mitred corner or the whole path (using Shift-select), you now have the ability to add Fillets to the corners.

From the context menu, two new commands exist, **Auto Mitre Selection** and **UnMitre Selection**. The **Auto Mitre Selection** option will use the settings from the **Technology** file, **Parameters**, **Auto Mitre** page.



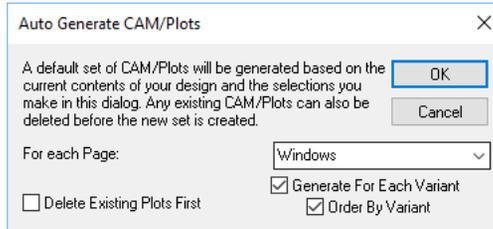
ODB++ Output includes Profile Cutouts

In the ODB++ format, the board outline is defined and drawn as a 'Profile'. The new switch **Include cutouts in the Profile** will enable the output to also apply Cutouts to the board (Profile). Some of the later versions of the ODB++ viewers have the ability to read cutouts in Profiles.



CAM Plots – Auto Generate Plot Ordered By Variants

Within the Schematic editor, in the **CAM Plots** option, there is a new switch in the **Auto Generate Plots** dialog to choose to generate plots and to **Order by Variant**.



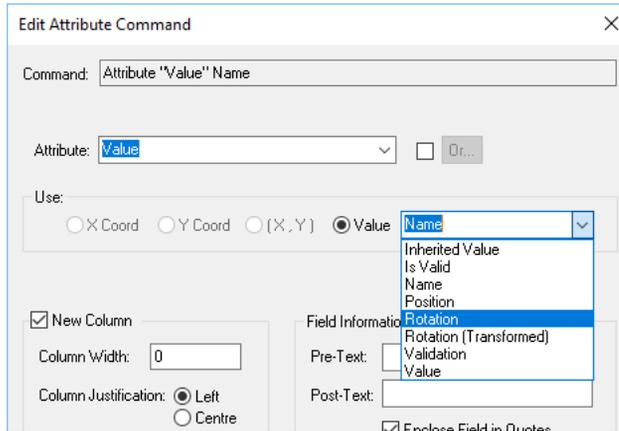
Report Maker

Rotation of Attribute Command

You can now report the **Rotation** of an **Attribute** position.

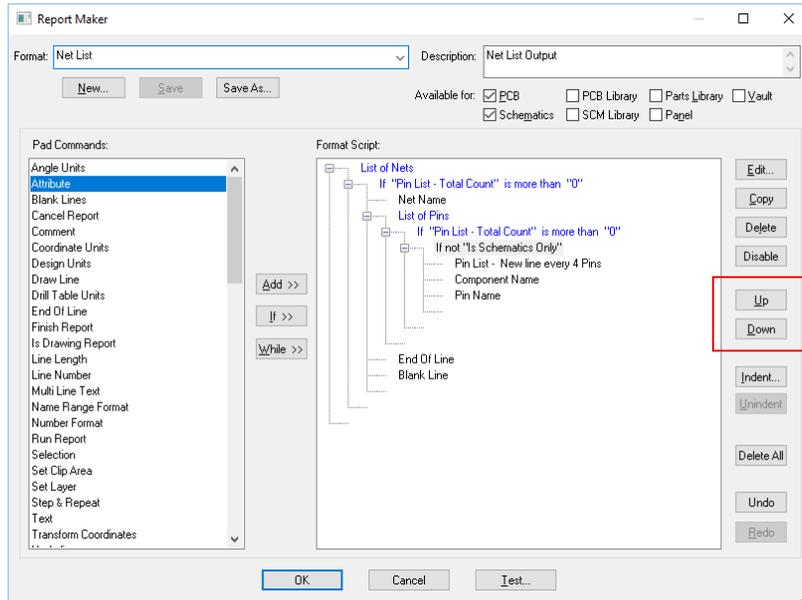
Rotation will output the rotation of the displayed attribute. This is the real rotation value, not the adjusted value for readable text. This will be reported in the rotation units and precision defined in the Units dialog.

Rotation (Transformed) will output the rotation of the displayed attribute with any design transformation applied also.



Indent & Unindent Commands

From the main **Report Maker** dialog, there are two new buttons on the right side of the pane for **Indent** and **Unindent**.

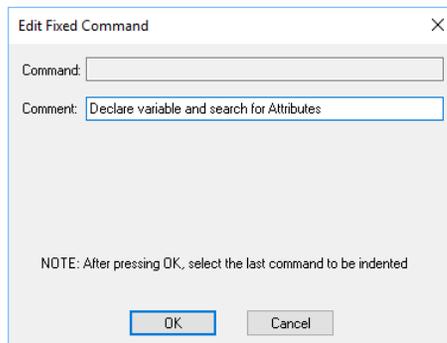


Indent

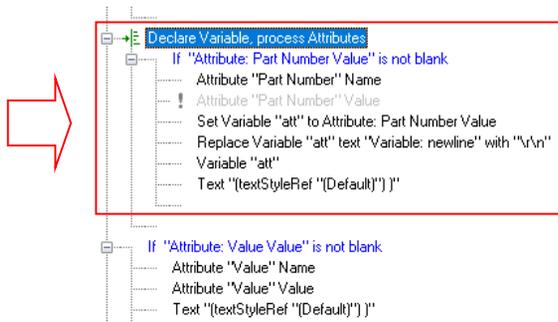
Use this to add indentation to a set commands. Indentation can be used for a couple of reasons:

- To add clarity to a portion of commands in a script. It might be a long IF statement that you need to indent to view clearer.
- By indenting a section of script, even commands outside of IF and ELSE statements, perhaps commands that follow the IF statements, such as End Of Line and Blank Line, you can then opt to Copy the commands or move them to another part of the script.

When the Indent button is pressed, after typing in a **Comment** into the dialog, you must select the portion of script required for indentation. This is an interactive command so drag the mouse up and down to select the portion of script required.



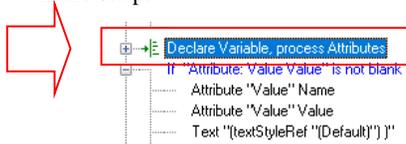
The comment is displayed in the script along with an indentation indicator.



This shows the indentation indicator:



The indent can then be 'closed' up to further aid clarity of your script. The Comment is still displayed in the script.



Unindent

Use this to remove the Indented status on a selected set of commands.

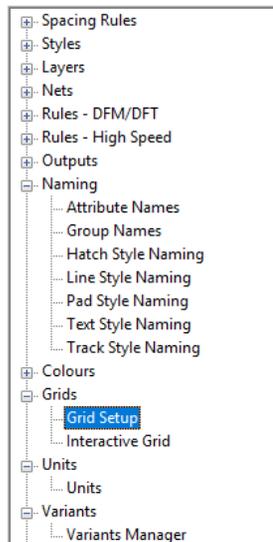
New In Version 10.0

The following documentation details features specifically for Version 10.0.

Technology Dialog Changes

Enhanced Technology Dialog

The Technology dialog has been further enhanced with the inclusion of **Colours, Units, Grids, Variants, Design Settings, Naming, Adjacent Nets, Advanced Router Parameters, (base) Router Parameters, Autoplace Parameters, Auto Gate/Pin Swap and Auto Mitre Parameters.**

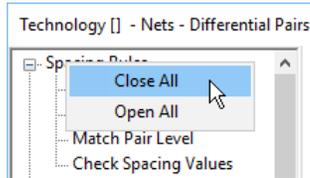


In addition to existing design facets, such as Colours and Units, version 10 also adds new functionality for auto-naming of style when required. Naming has been added under **Naming** for **Hatch Styles, Line Styles, Pad Styles, Text Styles and Track Styles**. See below for more details.

This means that many design Technology contents are now in one location.

Technology Dialog Tree Structure

You can now open and close all tree structures in the Technology dialog from the context menu.



New Style Naming pages for the Technology Dialog

Style Naming Dialog

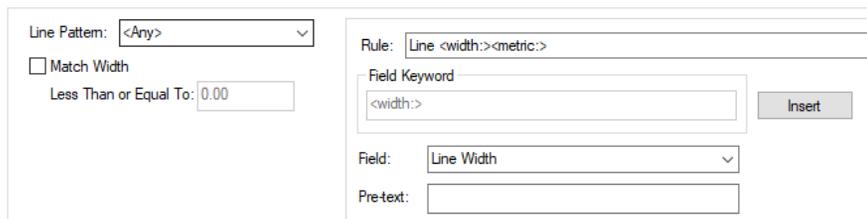
As a switchable option, you can opt to have Pulsonix automatically create style names for you. This means that, for example, when creating new pad styles, you could use the auto-naming option to use the IPC naming convention.

Style Naming is available for all **Styles** within the Technology.

Each page works like any other Rules page within the Technology except that generally speaking, you'd only ever have one or two rules for each naming style.

Line Pattern	Match Line Width	Line Width	Template String
Solid	<input type="checkbox"/>	0.00	Line <pattern> <width>

The edit portion of the dialog is used to define specific naming items, this varies for each item style. The right side of the edit pane defines the **Rule** that will be created. Once the rule has been defined using keywords selected from the **Field**: drop down list, use the **Insert** button to add it to the Rule.

A screenshot of the edit pane in the Style Naming dialog. On the left, there is a dropdown menu for "Line Pattern" set to "<Any>", a checkbox for "Match Width" which is unchecked, and a text field for "Less Than or Equal To:" containing "0.00". On the right, there is a "Rule:" field containing "Line <width><metric>", a "Field Keyword:" field containing "<width>", an "Insert" button, a "Field:" dropdown menu set to "Line Width", and a "Pre-text:" field.

Buttons on the Navigation bar under **Replace All**: enable you to add a rule to the rules pane replacing any existing rules. A button on all pages (**Basic Set**), is used to set up a basic rule for you to use. This can be edited to your own style if required. On the **Pad Style Naming** page, an extra button (**IPC Set**) is used to define a more complex string for the IPC naming rule.

Basic Set & IPC Set Buttons

On each of the pages is a button named **Basic Set**. On the **Pad Style Naming** page, there is an additional button for the **IPC Set**. These buttons are under the heading **Replace All**: and are used to create you a basic style naming rule tailored for each of the particular styles. As it says, using this button will add you one style naming rule to the grid and will remove all other rules defined. The rule won't be applied until the **Apply** or **OK** buttons are used at the bottom of the dialog.

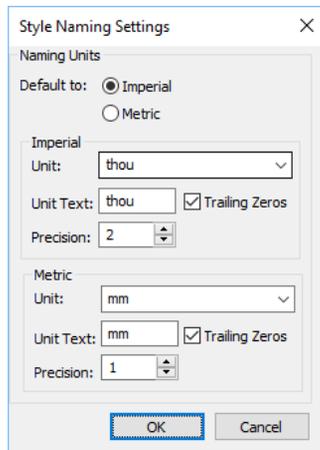


Note: When using the **IPC Set** button, this also sets up the **Settings** dialog as well as the rule definition. The IPC rule is a combination of both the rule and pad style/usage mappings.

Style Naming Settings Dialog

Each of the Style Naming dialogs has a **Settings** button. Each one is the same except for Pad Style Naming which has additional settings that can be used. The Settings dialog defines the **Units** and **Units Text** for the Style Naming. This is independent of the main Design Units dialog and is only used for Style Naming.

The standard Settings dialog looks like this:



The **Pad Style Naming Settings** dialog also enables additional **Mappings** for **Shapes** and **Pad Usage**.

This is used where specific styles require mapping so that can be correctly defined, the IPC naming style for example.

Pad Style Naming Settings

Shape Name Mappings

Shape	Label
Annulus	c
Bullet	d
Chamfered Rectangle	r
Diamond	u
Octagon	s
Oval	b
Pointer	u
Rectangle	r
Round	c
Rounded Rectangle	r
Square	s
Target	u
Triangle	u

Pad Use Mappings

Pads Intended Use	Label
Through Hole	
Surface Mount	
Via	v
Micro-via	v
Any	
All	

Naming Units

Default to: Imperial Metric

Imperial

Unit: thou

Unit Text: thou Trailing Zeros

Precision: 1

Decimal Character: .

Metric

Unit: mm

Unit Text: mm Trailing Zeros

Precision: 2

Decimal Character:

OK Cancel

Technology Styles Dialog

In addition to the Style Naming dialog for each of the styles, the actual **Style** page has been modified too to enable the naming.

Under each of the **Name:** fields for the style, there are now three additional radio buttons to define how the style will be **Named by:** There is a choice of **Typed** (manually named), **Rule** (using the new Style Naming rules for that style item) and **Template** (which is a hybrid, a combination of both Rule and Typed).

Mounting Hole Used:

Named by: Typed Rule Template

For Use By:

Through Hole Pads

Surface Mount Pads

Via

Shape: Round

Type: Round

Width: 3.8100

Length: 3.8100

Offset: 0.0000 0.0000

Drill:

Shape: Round

Width: 1.1430

Length: 0.0000

Inner Diameter:

Offset: 0.0000

Pad Exceptions By Layer Class & Inner Electrical layers

You can now create **Pad Exceptions** by **Layer Class** and for **Inner Electrical** layers. Both options are available when using the **By Layer** option and **Exception Type** field for a selected **Pad Style** in the **Technology** dialog.

The screenshot shows the 'Technology' dialog box with the following settings:

- Name: Mounting Hole
- Named by: Typed Rule Template
- Exception Type: **By Layer Class** (selected in the dropdown menu)
- Shape: Round (selected in the dropdown menu)
- Width: 3.810
- Length: 3.810
- Offset: 0.000 (left) 0.000 (right)

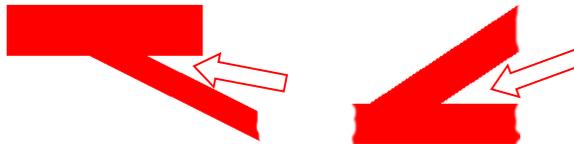
Acid Trap Rules

To facilitate additional functionality and checking, the **Design Spacings** page has been changed in the **Technology** to add spacing rules for **Acid Traps** under the **Tracks** tab. This checks for acid traps between Track segments and between Tracks and Pads.

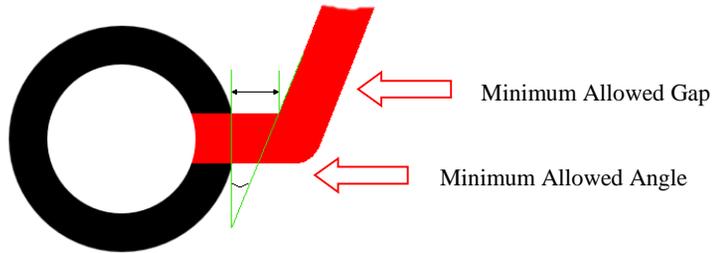
The screenshot shows the 'Additional Design Level Spacings and Rules' dialog box with the 'Track' tab selected. The 'Acid Traps' section has the following settings:

- Minimum Allowed Angle: 45.0
- Minimum Allowed Gap: 0.00

An acid trap is considered to be an acute angle created between design items, usually tracks and pads (as demonstrated below).



As well as the detection of acute angles (less than the **Minimum Allowed Angle** defined), the angle also represents calculated angles based on tracks closer than the gap between the pad and the offending segment. This is defined as the **Minimum Allowed Gap**.

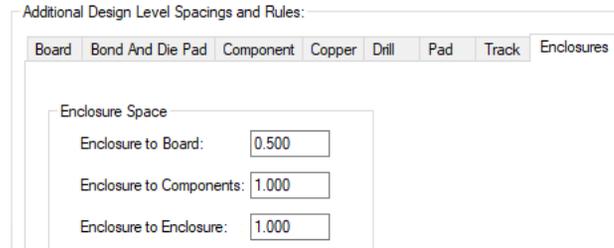


There is a new **Acid Trap** check box on the **DRC** dialog under the **Manufacturing** section.



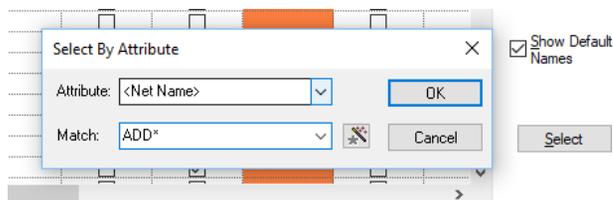
Enclosure Spacing Rules

To facilitate the changes within the **3D STEP Preview**, the **Design Spacings** page has been changed in the **Technology** to add spacing rules for **Enclosures**. *This is discussed below in more detail in the 3D STEP Changes section.*



Select Net Names by Attribute

There is a new **Select** button on the **Net Names** page of the **Technology** dialog.



30 Pulsonix Version 10.0 Update Notes

Once your **Match:** is chosen, pressing **OK** will make the matching selection in the **Net Names** grid.

	Name	Net Class	Type	Guard Space	Check Same Net	Use Own Colour	Colour	Display Connection	Attr
Y	+5V		Power	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	A		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	A1		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	A2		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD1		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD2		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD3		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD4		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD5		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD6		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADDR_DEC		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ALARM		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	

You can then use **Apply to selected cells** on the context menu, to, for example, set the **colour** of all nets with a matching attribute.

	Name	Net Class	Type	Guard Space	Check Same Net	Use Own Colour	Colour	Display Connection	Attr
Y	+5V		Power	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	A		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	A1		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	A2		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD1		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD2		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD3		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD4		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD5		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADD6		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ADDR_DEC		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	
Y	ALARM		Signal	0.00	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>	

Apply to entire Column

Apply to selected cells

Apply to all with same Net Class

New...

Copy...

Delete

Delete Unbound

Only Show Used Entries

Variant Manager now a grid Dialog

Along with the inclusion of the **Variants** option within the **Technology** dialog, this dialog has now been changed to be grid based similar in style to the other Technology pages.

	Current	Name	Description
	<input type="checkbox"/>	<Master Design>	Entire Contents of Master Design
Y	<input checked="" type="checkbox"/>	USA	120V variant
Y	<input type="checkbox"/>	GB	240V variant
Y	<input type="checkbox"/>	EU	220V variant

Name:

Used:

Description:

Global Settings

Show Variant on Status Bar

Not-Fitted Components: Display

Initially in a design, variants will not be in use. Press **New** to create the first variant. This will automatically create the **<Master Design>** entry in the Variants List. This entry is always at the top of the list and cannot be deleted while other variants exist.

Each press of the **New** button will add a new variant to the list. Select a variant from the list and check the **Current** button to make the one the current variant.

New Variants Button

Note, the **Use Variants** button has been removed. Adding a **New** variant will enable variants for designs with no variants currently defined.

New Adjacent Nets Rule

There is a new rule under **Technology, Rules – DFM/DFT** called **Adjacent Nets**. This allows you to set up rules to control what nets are checked against each other, check a net against a board outline and check a net against all items not on a net; that is, nets that match the rule and are within an optional tolerance. You may use this to check for nets that are inside the **Spacing Rules** defined but where you require knowing the closest item to the net being checked.

Options on this dialog control how many warnings are generated and whether the set of nets is checked as a whole, or each net checked individually.

Warnings are listed separately in the **Errors bar** and cannot be locked.

Check Nets		Against Nets		Check Against Board	Check On			Adjacent Nets			
Attribute Name	Match Value	Attribute Name	Match Value		Side	Layer	Area	Individual Nets	Within Tolerance	Tolerance	Max Warnings
<Net Name>	*	<Net Name>	*	<input type="checkbox"/>				<input type="checkbox"/>	<input type="checkbox"/>	<Unrestricted>	1

Check Nets:

Attribute: <Net Name>

Match: *

Against Nets:

Attribute: <Net Name>

Match: *

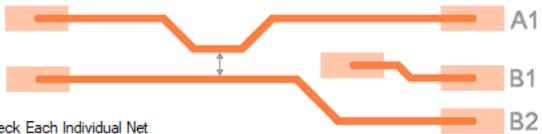
On Layers:

Side: <Any>

or Layer: [dropdown]

Within Areas: [dropdown]

Check Against Board



Check Each Individual Net

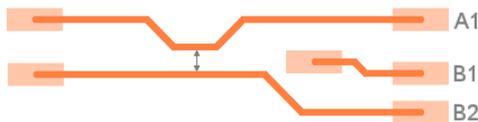
Within Tolerance: <Unrestricted>

Maximum Warnings For Each Check: 1

With the flexibility of Pulsonix, the rule can be defined very broadly, such as all nets to all nets, or it can be defined more specifically, such as Net Name XXX against Net YYYY on Layer X.

Additional rules can be also defined to refine the rule depending on your requirement.

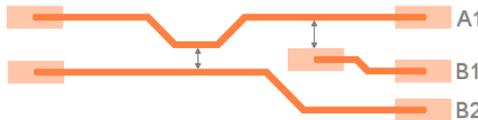
The general rule without the extra check boxes selected will select use a basic adjacency rule.



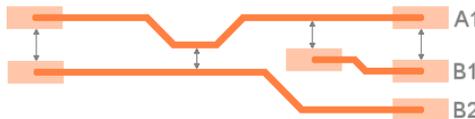
Checking the **Check Against Board** check box enables checking between nets and the board outline.



Checking the **Check Each Individual Net** will check every net pair combination in the two sets, and not each set of nets as a whole.



Selecting the **Within Tolerance** will define a tolerance value, so that only near points less than that tolerance are reported.



Maximum Warnings For Each Net - this check box enables you to define the maximum number of warnings that each net can produce. Enter a value between 1 and 10. A large value may cover your design with warnings, meaning they are difficult to resolve. If there are many warnings, you should check your spacing rules.

Adjacent Nets Checks in DRC Dialog

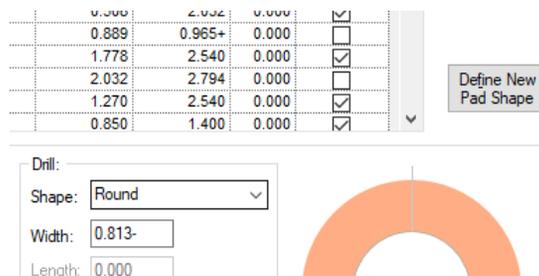
A new check in **Nets** section of **DRC** dialog has been added for **Adjacency Nets**. This reports the closest points of adjacent nets with a warning marker. This check runs in its own thread.

Adjacent Nets in Report Maker – List of Errors

Report maker can detect the **Adjacent Nets** errors in a **List of Errors** command by using the **Is Warning** command. You can report the net names using the new **Item1Name** and **Item2Name** commands.

Define Pad Shapes now on Pad Styles dialog

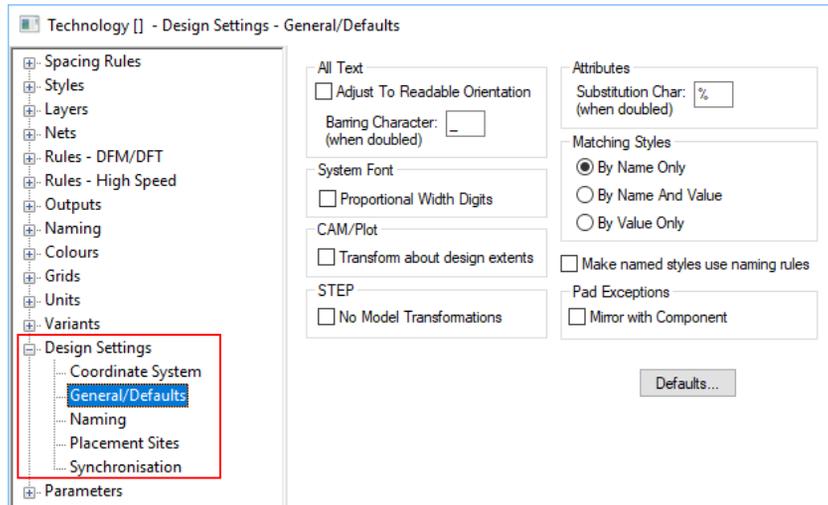
As well as the **Define Pad Styles** option on the **Setup** menu, it has now been included as a button in the **Technology** dialog under **Pad Styles**.



Design Settings now in Technology Dialog

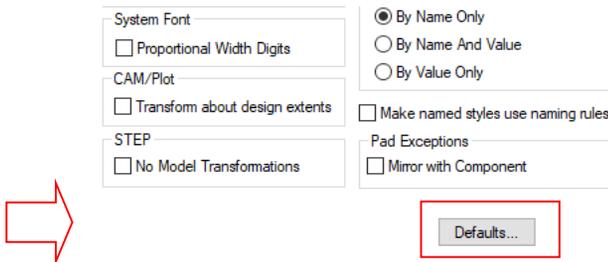
Design Settings Menu

Design Settings are now included as part of the **Technology** dialog. They are listed under their own menu item, **Design Settings**.



Defaults Pages

In the **Technology** dialog, **Design Settings** menu, **Defaults** pages are now available on the **Defaults** button which is on the **General/Defaults** page.



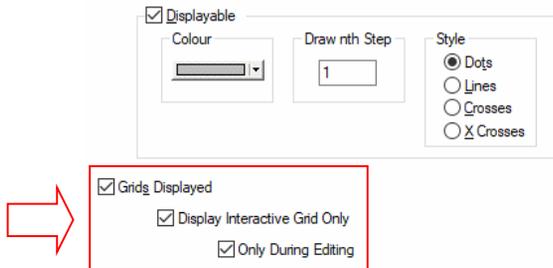
Grids now in Technology Dialog

Grids Menu

Grids are now included as part of the **Technology** dialog. In the **Technology**, **Grids** menu, **Current Interactive Grid** is now replaced with **Interactive Grid**.

Grids Displayed Switches

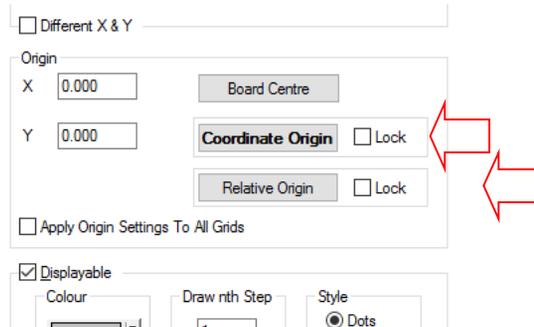
There is a new switch in the Grids dialog, **Only During Editing**.



With **Grids Displayed** checked, when **Display Interactive Grids Only** is checked, the new **Only During Editing** switch decides what happens when in select mode. If **Only During Editing** is checked then when in **Select** mode will display all grids that are marked as **Displayable** (as it did in previous versions). If unchecked, select mode will display the interactive grid.

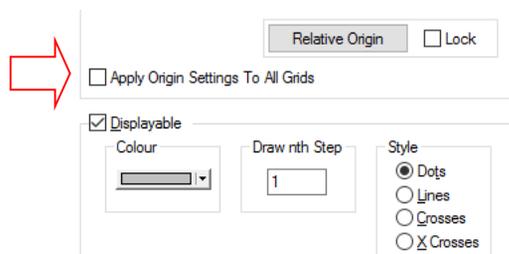
Locked Grid Origins

The ability to lock a grid to an origin of a particular type has been added. With the **Lock** check box selected, if the origin is moved, the grid will automatically update to be relative to it still.



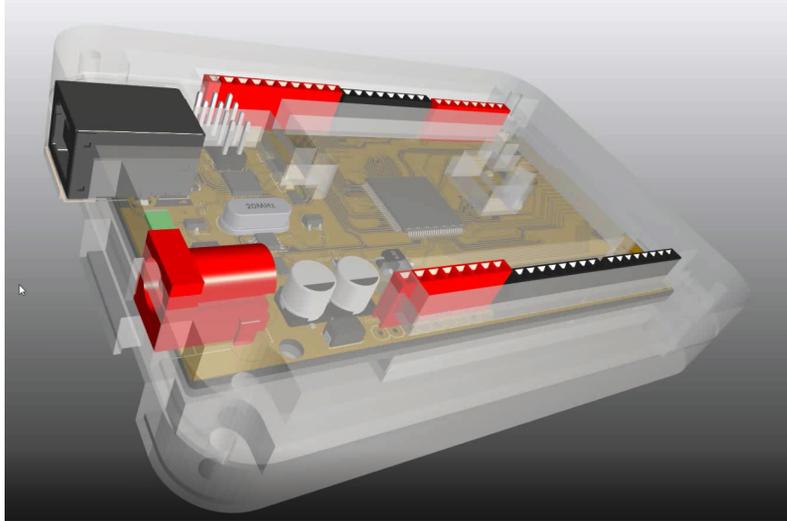
Apply Origin To All Grids

A new check box labelled **Apply Origin Settings To All Grids** has been added under **Grid Setup** dialog. All grids will take on the settings from a grid the check box is applied to, e.g. Working grid can use a **Locked Coordinate** that is then applied to all other grids.



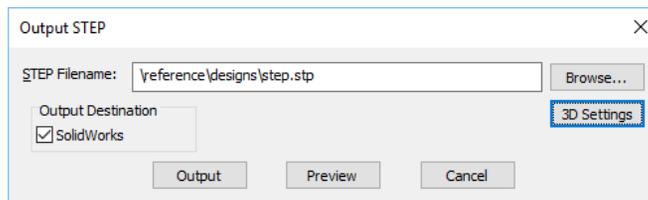
3D STEP Preview Changes

The **3D STEP** engine (STEP Preview), interaction, preview and environment within Pulsonix has been significantly reworked for Pulsonix 10.

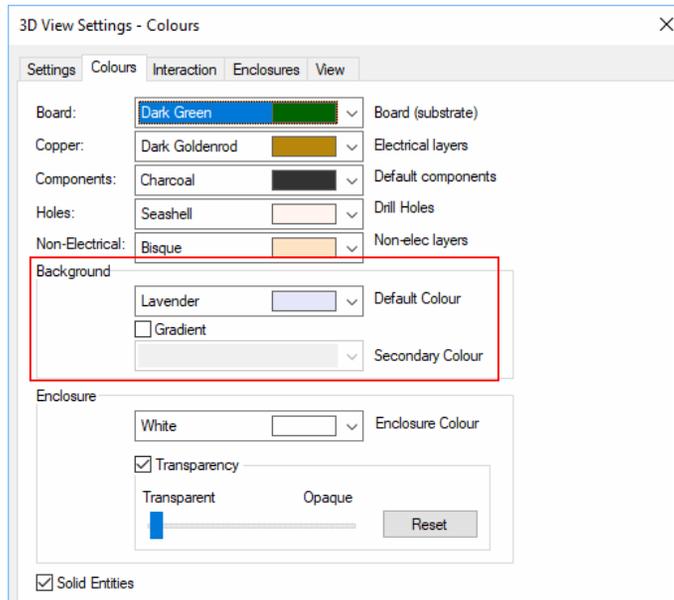


Gradient Background (switchable between Solid and Gradient)

The background can now be switched from a single solid colour to a gradient background between two colours defined in the **3D View Settings** dialog under **Colours**. You can setup the colours prior to viewing your 3D Preview from the **Output** menu, **OUTPUT STEP** dialog and **3D Settings, Colours** tab, or from within the STEP Preview from **Setup** menu, **3D Settings, Colours** tab.



Both the **Background** colour and **Gradient** colours can be defined. Without the Gradient check box enabled, just the Background colour chosen will be used as a solid colour. With the Gradient check box selected, the background will be graduated between the background colour and the Gradient colour from the top to the bottom of the screen.



Improved Visualisation of the STEP Preview

As a visualisation improvement generally for the 3D Preview, anti-alias rendering on models has been introduced. This provides smoother edges and a more realistic look to the design. In addition, the lighting of the design has been improved to add to the realism.

Print option in STEP Preview

The **Print** option has been removed from the **File** menu and a **Save As Image** option has been added instead.

Toolbar with Icons of Pre-defined 3D Views

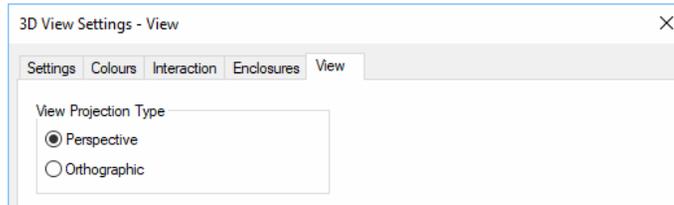
Within the 3D STEP Preview, there is now a toolbar containing pre-defined 3D Views available for selection. Views have been provided (moving from left to right across the toolbar) for **Axonometric**, **Front**, **Back**, **Left**, **Right**, **Top** and **Bottom**.



For the perspectives used in **Axonometric** view, please see below under *Perspective and Orthographic Views*.

Perspective and Orthographic Views

For the **Axometric** view, you can switch between **Perspective** and **Orthographic Views** depending on your preference. This can be changed from the **Setup** menu in the 3D environment and **View** tab or from the PCB design, **Output** menu, **STEP**, **3D Settings**, **View** tab.



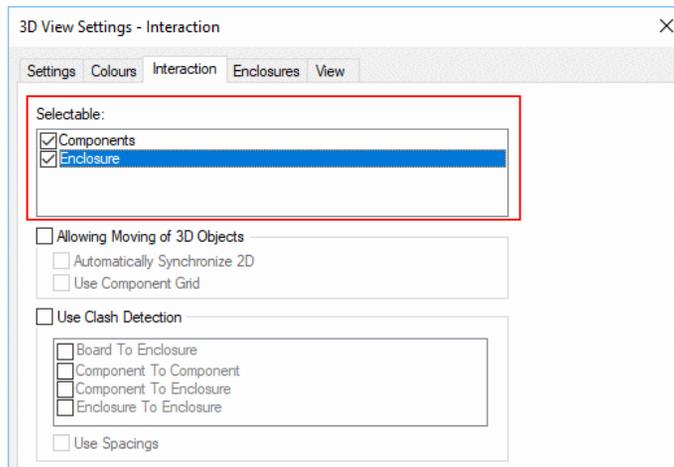
Note: when the perspective is changed, this is only applied when you next run the STEP Preview, it is not changed for this Preview session. A warning to say this is issued when the perspective is changed.

Rotation of Design in STEP Preview now on Right Hand Mouse button

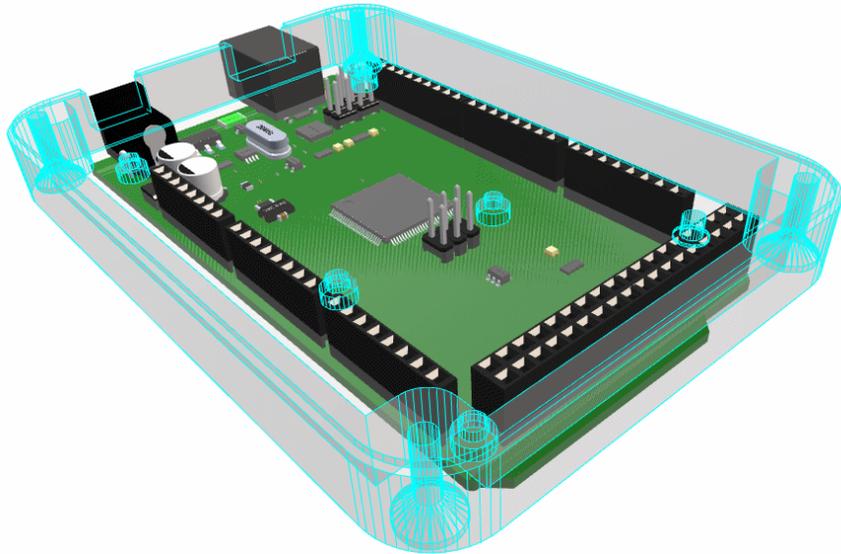
Rotation of the design in the STEP Preview is now done using the **right-hand mouse button with the Ctrl button** also pressed. Previously, this was done using the left-hand mouse button. The movement of this command to the right-hand mouse frees up the ability to select items using the left-hand mouse (*this is discussed below in more detail*).

Selection of items in STEP Preview on Left Hand Mouse button

Using the **left-hand mouse button**, you can select both components and enclosures when they have been checked as selectable from the **Setup** menu in the 3D environment and **Interaction** tab or from the PCB design, **Output** menu, **STEP**, **3D Settings**, **Interaction** tab.

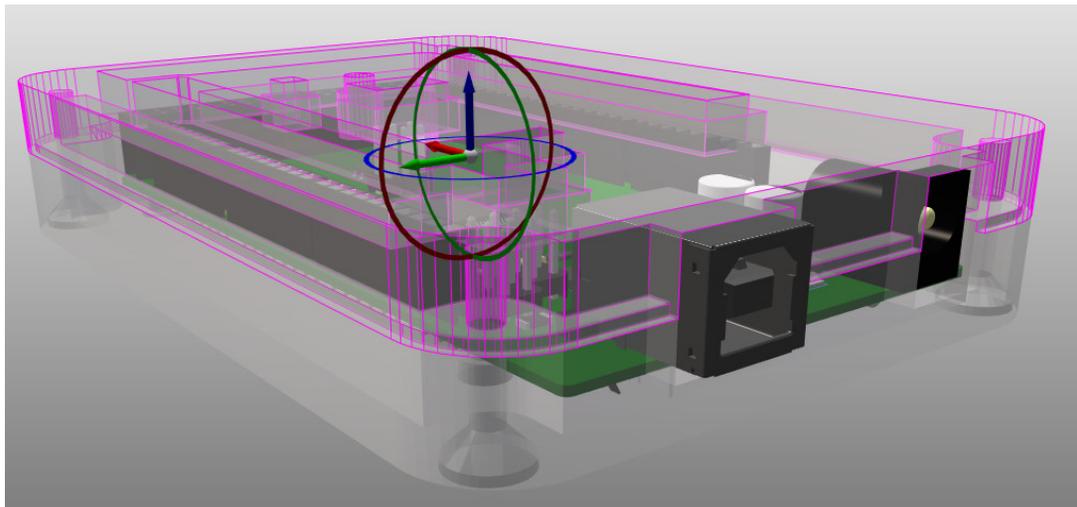


With this option enabled, if the mouse is moved over the either a Component or enclosure within the STEP Preview a wire frame selection colour will be added to the item.



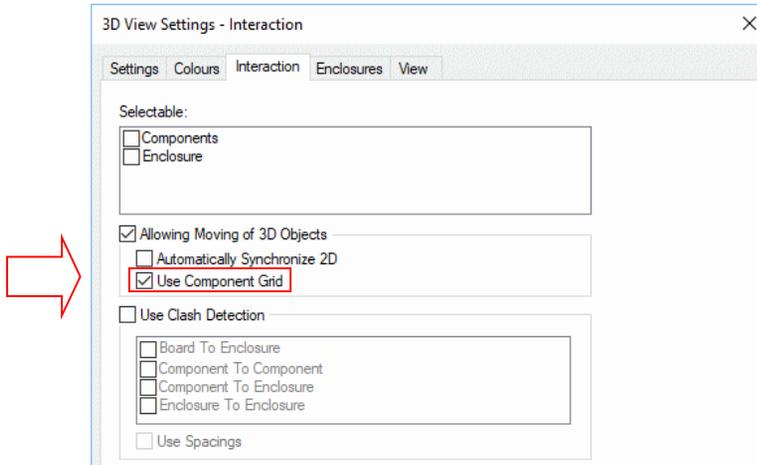
If you use a single click of the mouse over the model, the model movement axes are displayed indicating the movement options available to you. This is displayed if the Component or Enclosure is marked as **Selectable** on the **Settings, Interaction** dialog.

Components will be able to move in an X and Y direction, whereas enclosures can be moved in X, Y, Z and Rotational axes.



Move Component on Grid in STEP Preview

There are two settings for moving Components in the 3D STEP Preview; on the **Component Grid** and without. The default mode is for the movement to be on the **Component Grid** (as defined in the PCB design). It is recommended not to change this setting.



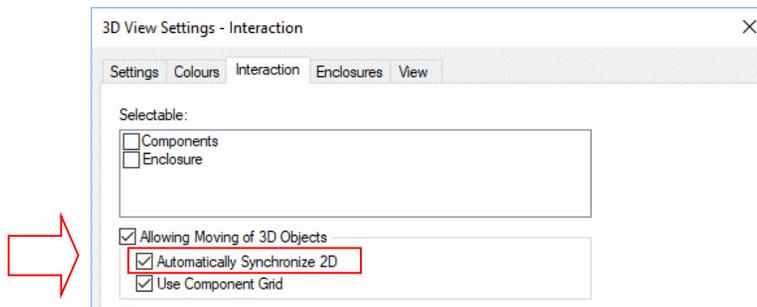
Moving Components in the 3D Preview will use the Component Grid defined in the PCB. This is dynamic, meaning that if the grid is changed in the PCB (with the 3D Preview still enabled in another window), the grid step will change in the 3D Preview to match. Likewise, if you change the PCB grid to use another grid, such as the <Working> grid instead of the <Component> grid, this will now also be reflected in the 3D Preview and be used.

As an application note, you may find it easier to move Components within the Top view perspective rather than a 3D view.

Automatically Synchronise 2D PCB from 3D

You can opt to have Component and the Enclosure positions that are changed in the 3D Preview automatically made back in the 2D PCB view.

To do this, select the **Automatically Synchronise 2D** check box on the **Interaction** tab of the **3D Settings** dialog. By default, this will be enabled.



Working in both 2D and 3D Environments

If you intend working in both the 2D PCB environment and 3D Preview at the same time, and moving components around you should be aware of the following:

- If Components are moved in the PCB, then the 3D Preview must be regenerated (close it, then regenerate using the **Output** menu and **STEP Preview**).

40 Pulsonix Version 10.0 Update Notes

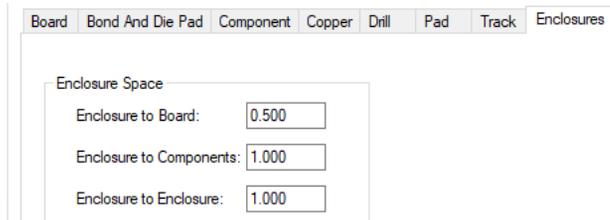
- If Components are moved in the 3D Preview, then provided the **Automatically Synchronise 2D** check box on the **Interaction** tab of the **3D Settings** dialog is checked, then the component move will be automatically reflected in the PCB without doing anything else.

If adding Enclosure STEP models within the STEP Preview environment, these are also back annotated to the PCB design.

Enclosure Spacing Rules

New **Design Spacing Rules** are available for **Enclosures**. These are defined in the PCB design and transferred to the STEP Preview when the dialog is opened.

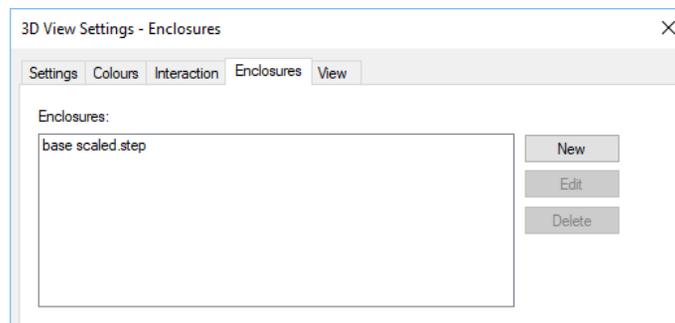
Spacing rules available for **Enclosure to Board**, **Enclosure to Components** and **Enclosure to Enclosure** will be used when the design is transferred into the **3D STEP Preview** and used for clash reporting.



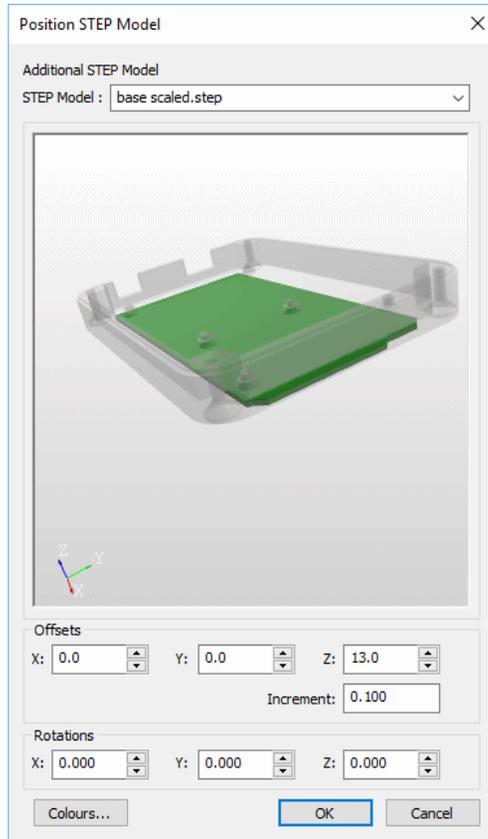
See below under *Clash Detection within the 3D Environment* for how the Spacings are used.

Adding Mechanical Enclosures

Mechanical enclosures in STEP format can be included along with the Board and Components (using STEP models). There is a new **Enclosures** page in the **3D View Settings** dialog of the **Setup** menu when within the STEP Preview to add Enclosures or other STEP models that are not defined as Components to the STEP Preview. You may also add more than one enclosure to the design for use in the Preview. This is also available from the PCB design, **Output** menu, **STEP**, **3D Settings**, **Enclosures** tab.



On selection of the **New** button, the **Position STEP Model** dialog is displayed. This will allow you to add a new or additional STEP models:



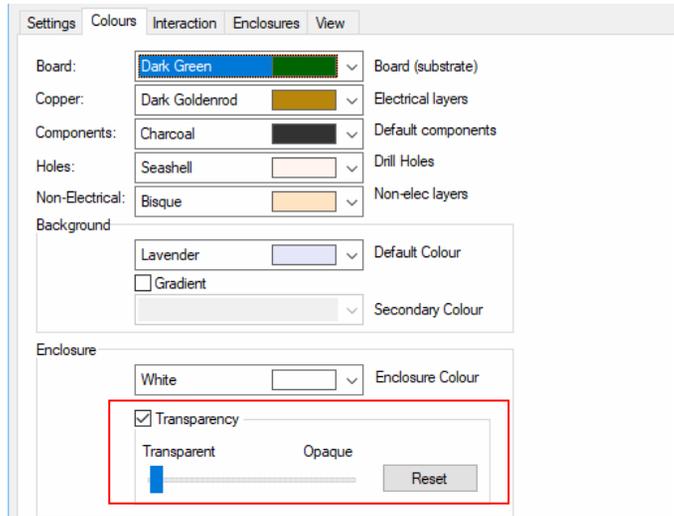
If you select a model from the list, the **Edit** button will allow you to edit its position.

The **Transparency** of these STEP models is taken from the **3D Settings, Colours** dialog. Colours used are available on the Colours button which are replicated from the **3D Settings, Colours** dialog.

Transparency Setting for Enclosures

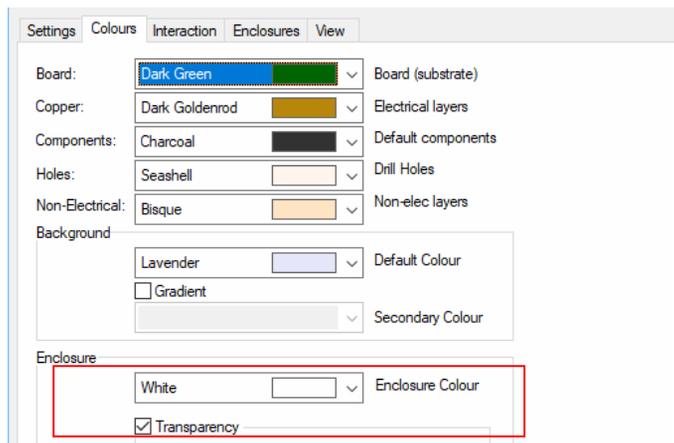
With the introduction of Enclosures and additional STEP models that are not Component related, you can set the **Enclosure Transparency** so that when overlaid on a board with Components, you can see the contents inside the enclosure. This is set on the **3D View Settings** dialog under **Colours**.

42 Pulsonix Version 10.0 Update Notes



Colour Setting for Enclosures

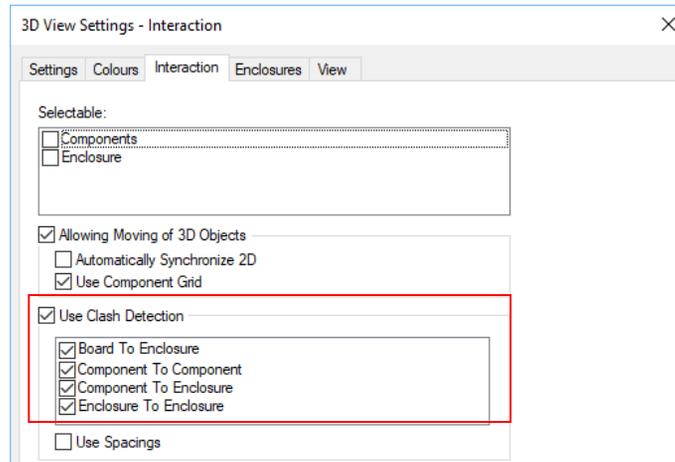
Where enclosures are used and they have no colour defined of their own (this is normal as some are defined as neutral), there is a default colour setting that will be used by Pulsonix. On the **3D View Settings** dialog and **Colours**, there is an **Enclosure** colour that can be set.



Clash Detection within the 3D Environment

To use the **Clash Detection** feature within the STEP Preview, you must first enable it, then enable the options you wish to detect. These options are available in **3D View Settings** dialog and **Interaction** under **Use Clash Detection**.

If using the STEP Preview as simply a preview of the board, you may wish to switch off both the **Selectable** setting of **Components** and **Enclosures** and the **Use Clash Detection** options.



See below under *Using The Clash Detection Option* for information about using this option.

Using the Clash Detection Option

Once the **Clash Detection** option has been enabled (in **3D Settings, Interaction, Use Clash Detection**, see above), you can detect clashes using one of two options:

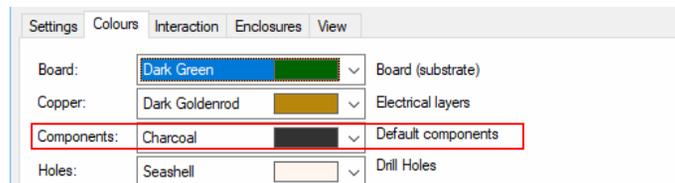
- Interactively move Components or Enclosures
- Or you can use the **Detect Clashes** option from the **Tools** menu

Once clashes are detected, they are highlighted in red.

To un-highlight clashes, select the **Remove Clash Highlights** from the **Tools** menu.

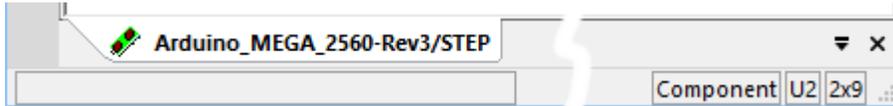
Colouring for non-coloured STEP Models

Where STEP models used for Components have not had their colour defined in the model, which some don't, Pulsonix will add a colour for you. Choose the **Components** colour under the **3D Settings** and **Colours** tab.



Properties of Selected design items shown on Status Bar

For selected Components in the design, basic **Properties (Component Name and Part Name)** will be displayed on the **Status Bar**. It also shows you that a Component has been selected.

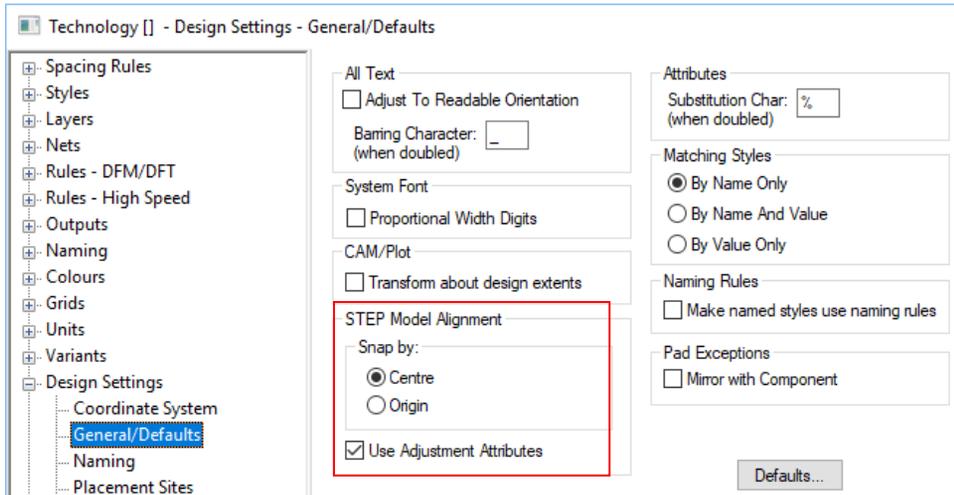


Speed Increase of STEP file Generation through Multithreading

With the introduction of multi-threading capabilities within the STEP file generation process, with drill holes selected, there is a significant speed increase. On large files tested, a speed increase of between 80-95% has been recorded. Obviously, the content of the design and the entities being exported will make a difference to the speed.

STEP Model to Footprint Alignment

The STEP option in the **Design Settings** dialog, **General/Default** page has changed from where it showed **STEP** and **No Model Transformations** to this below:



Checking the 9.1 **No Model Transformations** option had two effects:

- Ignored the <STEP Offsets> and <STEP Rotations> attributes attached to Parts or Footprints when aligning a STEP model to a footprint.
- Stopped the automatic alignment of a STEP model to a Footprint by using bounding box centres (or Model Placement Areas)

In Pulsonix 10, the **Snap by:** values will have the following effect:

- **Centres** - will align the Footprint and the model using bounding box centres in the same way as in Pulsonix 9.1 without the **No Model Transformations** check box selected.

- **Origins** - will align the origin of the Footprint with the (0,0) in the model without reference to bounding boxes or areas in the same way as in Pulsonix 9.1 with the **No Model Transformations** check box selected.

The **Use Adjustment Attributes** option will effectively do the same as above in 9.1 and ignore the <STEP Offsets> and <STEP Rotations> attributes, regards of the **Snap by:** value.

3D Package Built-in Attribute

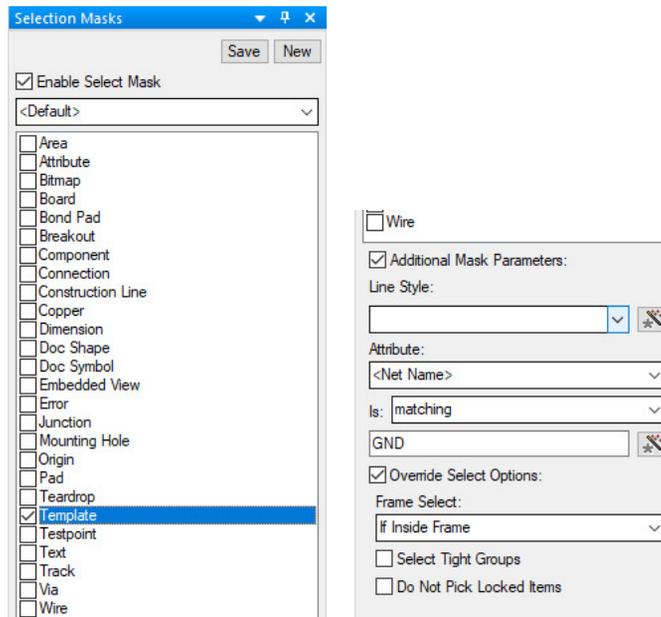
The <3D Package> attribute is used to override the normal method of finding a 3D package based on the footprint name. There has been a subtle change made for V10 is to detect an empty value for this attribute indicating that there should be no 3D component displayed on the 3D view. This allows you to suppress any attempt to display a 3D image for particular components such as edge connectors when they aren't required.

Note: when the 3D Viewer is used under normal circumstances, unless specifically required, the <3D Package> attribute will not be used on a Component or Footprint as it is automatically mapped from the footprint name.

Selection Mask

The new **Selection Mask** toolbar allows you to choose design items that are selectable within the design and save the mask for subsequent reuse. This is available in both the Schematic and PCB design editors.

This new option is available from the **View** menu and from the custom menu enabled by right clicking on any of the Pulsonix framework (enabled the same way as the **Design Browser, Part Browser, World View** etc.). It can also be enabled by right clicking on any of the existing open Browser windows and selecting it from the context menu.



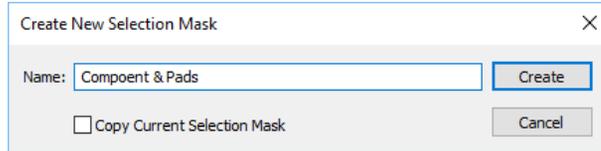
Using the browser dialog

You can use the Selection Mask on a task by task basis or you may prefer to create a selection mask and save it for subsequent use.

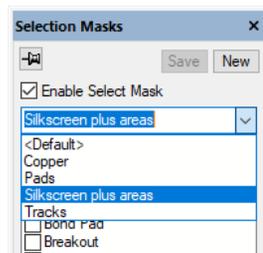
You can create a list of selection masks that can be defined and saved. You might create a number of selection masks for all Track and Pads for example.

To activate the Selection Mask and start to use it, select the **Enable Select Masking** check box. Without this selected, nothing happens. Initially, the <Default> selection mask is shown. This is the selection of check boxes in the list as currently set. If you change the selections and press the **Save** button, this then becomes the new <Default> selection. However, you may wish to be more specific about your choices and to reuse them.

You can save multiple named selections by pressing the **New** button. When this button is pressed, the **Create New Selection Mask** dialog is displayed. When the **Create** button is pressed, the selection mask is saved. Using the **Copy Current Selection Mask** will enable you to make the selections, then save them. You could also use this to make slight modifications to an existing selection mask and save it to another named selection.

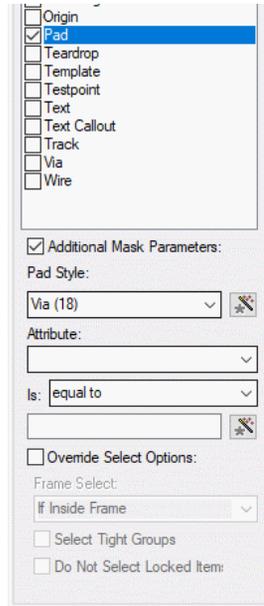


The drop down list on the Selection Mask dialog displays all previously saved Selection Masks



Check box options on the Selection Mask Dialog

There are additional filters to enable refinement of the selection. With **Pads** selected, selecting the **Additional Mask Parameters** enables the Pad Style drop down list from where you can choose a pad style. This can be a specific pad style or can be a wildcard name. The Wildcard wizard helps you create the name.



With a Pad style selected, you can then only choose pads that match that style.

Updated User Interface

4K Graphics Support

New icons have been created to enable 4K monitors to be supported. The new icons are scalable and will be automatically resized when Pulsonix detects the use of monitors that require the new icons.

As part of the 4K update, the use of graphics within Technology dialogs and other dialogs/wizards has also been resolved.

Simplified Default Toolbars

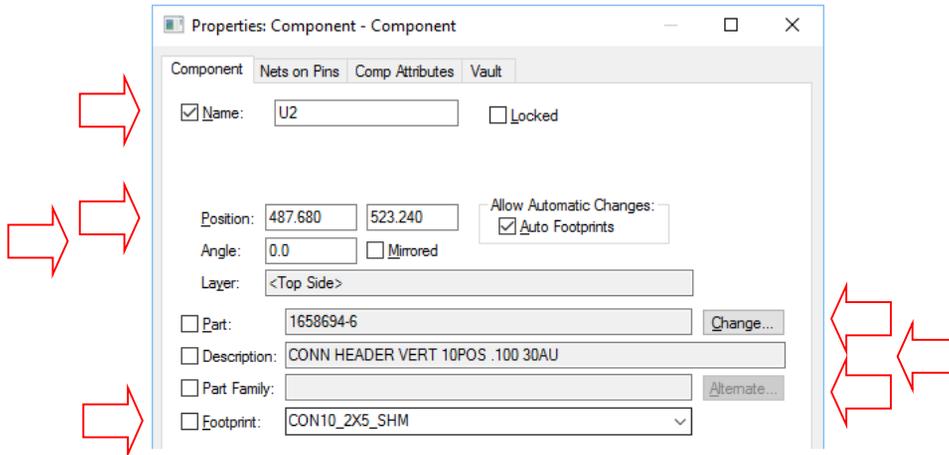
As part of the improved user experience, the default toolbars have been simplified by removing non-essential icons. If you have migrated from an older version of Pulsonix (9.1, 9.0 or previous), you will need to reset the toolbar icons using the **Customise** option (from the **Tools** menu), **Toolbars** tab and **Reset All** button.

You will also see that menus have been decluttered with the removal of non-essential icons. These icons are still available for use within a customised toolbar if required.

More Readability for Read Only Edit Boxes

Non-editable and read only edit boxes now have a set disabled colour to differentiate from edit boxes that can be changed and those that can't. This makes their editable status more obvious when using high resolution monitors.

In the example dialog below, it can be clearly seen that entries such as **Name**., **Position**: and **Angle**: can be edited, whereas the **Part Name**, **Description** and **Part Family** can't.

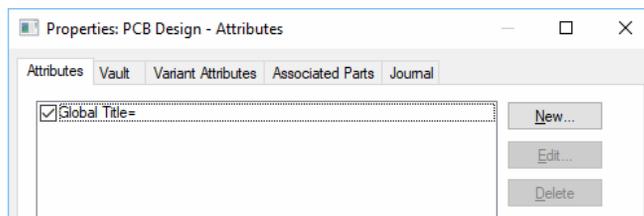


Design Attributes in Variants

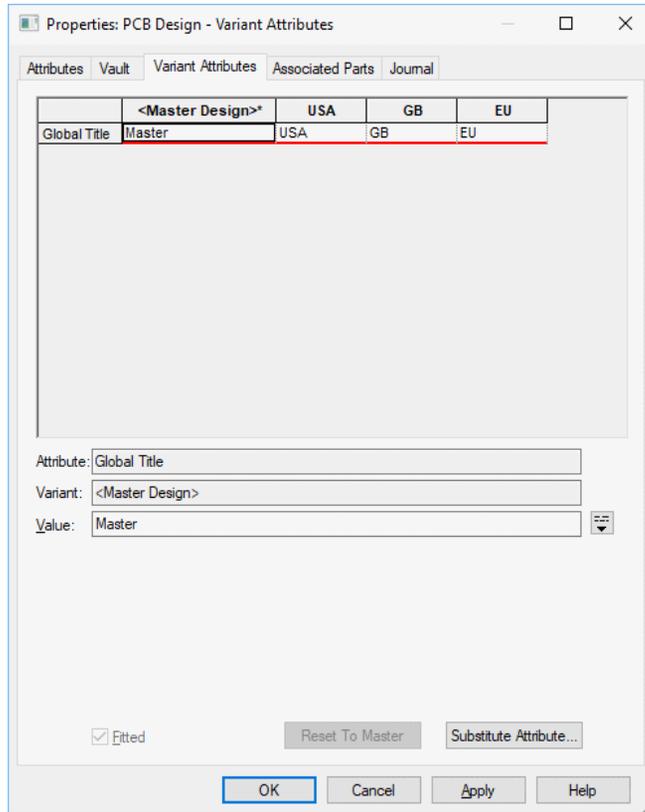
Design level Variant Attributes can now have different values in each variant (as well as a fitted state). These are accessed through the **Edit** menu and **Design Properties**.

	<Master Design>*	USA	GB	EU
Global Title	Master	USA	GB	EU

The **Design Attribute** must first exist, this is defined on the **Attributes** page of **Design Properties**.



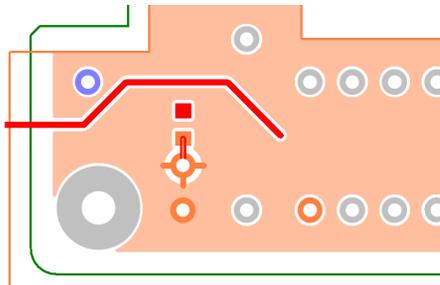
Each Variant instance for the attribute, in our example above, **Global Title**, can be edited to display a different value.



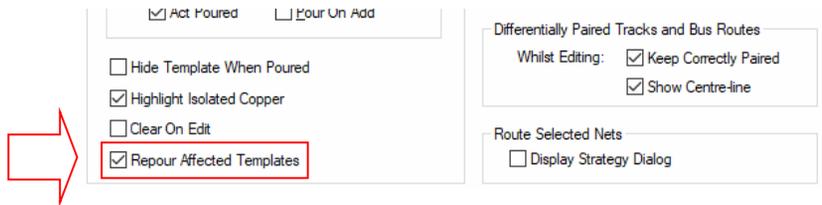
For each attribute it can also have a different **Fitted** status

Interactive Copper Pour Healing

The ability to ‘heal’ copper pour areas dynamically after edit has been added. The healing process will only heal poured regions when damage has occurred due to the edit. This may be caused by a track being moved into it or a pad being dropped in an area of poured copper.



This feature can be switched on or off using the **Interaction** dialog in **Options**. By default, it will be enabled.



Changes to Manual Routing Modes

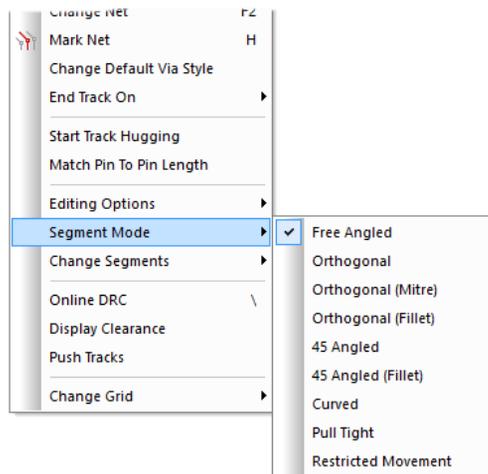
Segment Modes

The existing **Segment Mode** menu items used for manual routing and Differential Pair routing have been adjusted to make them more obvious. All modes now displayed on the menu were already available previously but have now been expanded into individual 'modes' with a more verbose description.

The previous **Orthogonal Mitre** and **Curved Mitre** sub-options for the **Orthogonal Mode** have been replaced with new segment modes **Orthogonal (Mitre)** and **Orthogonal (Fillet)**.

Also, the **Angled** segment mode has been renamed to **45 Angled**, along with a new mode called **45 Angled (Fillet)** that adds a fillet to every 45 degree corner.

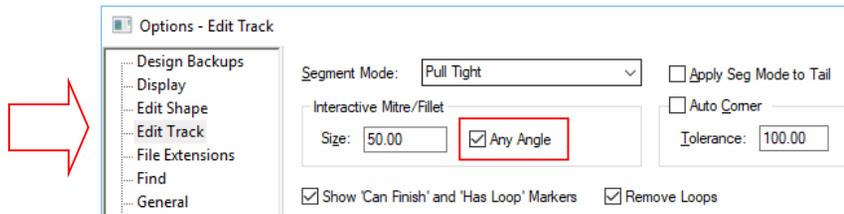
Fillet modes are different to Curved in that they provide a smooth transition between angled segments.



These modes are also available for use during **Differential Pair** routing if you have the Interactive High Speed option on your license.

Edit Mitre Mode

There is a new **Any Angle** check box in **Interactive Mitre/Fillet** section of the **Edit Track** and **Edit Shape** options pages.



Checking this box to allow the **Edit Mitre** option (often assigned to Double-click on a corner) to mitre or fillet between straight lines at any angle between them.

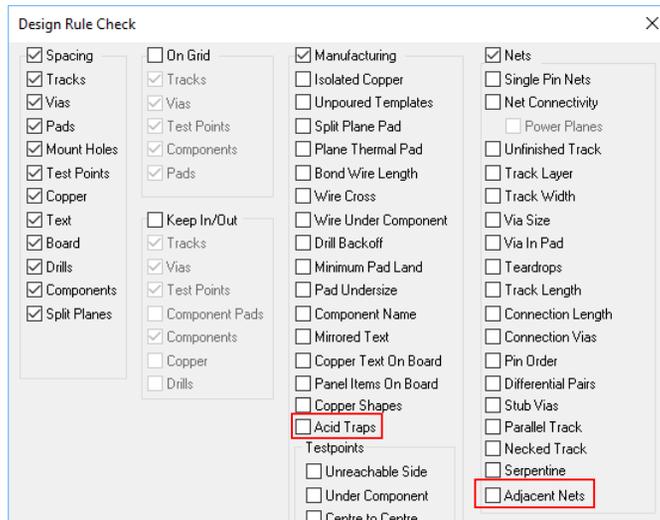


Leave it unchecked to only allow mitres between lines orthogonal and perpendicular to each other. This does not affect the Mitre Corner or Auto Mitre options.

Design Rules Checks Dialog Changes

New DRC Checks - Acid Traps and Adjacent Nets

Two new checks have been introduced for **Acid Traps** and **Adjacent Nets**.



52 Pulsonix Version 10.0 Update Notes

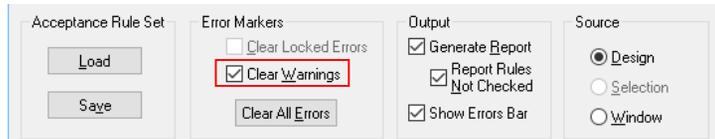
Both options are described in more detail under function headings, but in summary:

Acid Traps is a check defined in the **Design Spacing Rule** dialog under the **Tracks** tab. This rule defines both the distance of a track to a pad and checks for acute angles.

A new check in the **Nets** section of **DRC** dialog has been added for **Adjacency Nets**. This reports the closest points of adjacent nets with a warning marker. The adjacency rules are defined in the **Technology** dialog under **Rules – DFM/DFT**.

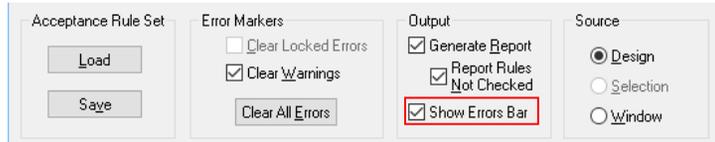
Clear Warnings Option

As some of the new options such as **Adjacent Nets**, report warnings and not errors, the DRC dialog has a check box to enable you to clear existing warnings (**Clear Warnings**) when DRC is run.



Automatically Open DRC Errors Bar Switch

An extra button has been added to the **DRC** dialog to automatically open the **DRC Errors Bar**.



Attribute Editor Changes

Footprint Name in Edit Attribute Dialog

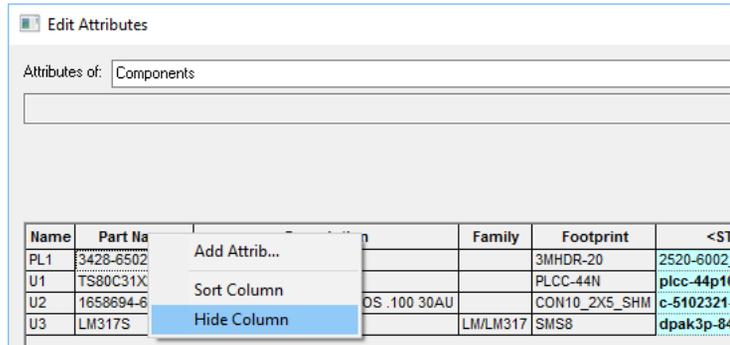
The **Attribute Editor** has been made more consistent across all areas where it is used; in the Schematic and PCB design editors and within the Library Manager on the Parts page. All column fields such as **Part Name**, **Description**, **Family**, and **Footprint Name** have been made available where they were previously missing.

The screenshot shows the 'Edit Attributes' dialog box. At the top, it says 'Attributes of: Components' and 'Capacitance = 106'. Below is a table with columns: Name, Part Name, Description, Family, Footprint, Capacitance, Category, and Code. The table contains several rows of component data.

Name	Part Name	Description	Family	Footprint	Capacitance	Category	Code
AC	22-27-2021	KK Header+Lock 2.5mm Pitch	CONN/22-27-2	MOLEX 22-27-2021		Connector/Header	
BELL	22-27-2021	KK Header+Lock 2.5mm Pitch	CONN/22-27-2	MOLEX 22-27-2021		Connector/Header	
C1	SR305C106KAA	Ceramic Radial Lead Epoxy MLCC X7R AVX RoHS	C/SR30	SR30	106		
C2	SR301C105KAA	Ceramic Radial Lead Epoxy MLCC X7R AVX RoHS	C/SR30	SR30	105		
C3	SR212A220KAR	Ceramic Radial Lead Epoxy MLCC COG AVX RoHS	C/SR21	SR21	220		
C4	SR212A220KAR	Ceramic Radial Lead Epoxy MLCC COG AVX RoHS	C/SR21	SR21	220		
C5	489D106X0010C1	CAP 489D 16V TANT CASE C	C/489D-C	BEAD 0.6mm		Capacitor/Radial Lead/S	

Hide Inbuilt Columns

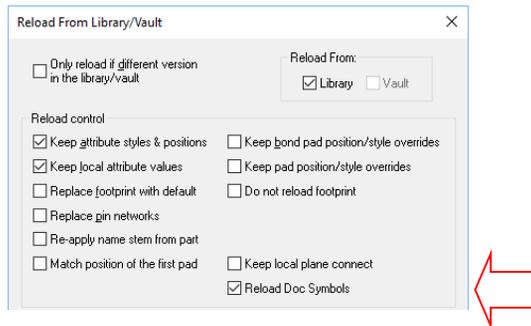
You can now quickly hide the inbuilt columns (except the Name column). The **Hide Column** option is available from the context menu on the column header. To show columns again, choose the field required from using the **Column Visib** button on the controls panel.



In addition, the Part Description and other standard fields are now on all Part based grids.

Reload All – Do Not Reload Doc Symbols

You can now choose not to **Reload Doc Symbols** when doing **Reload All**. This is useful as Doc Symbols are not relevant to the Synchronise Designs process.

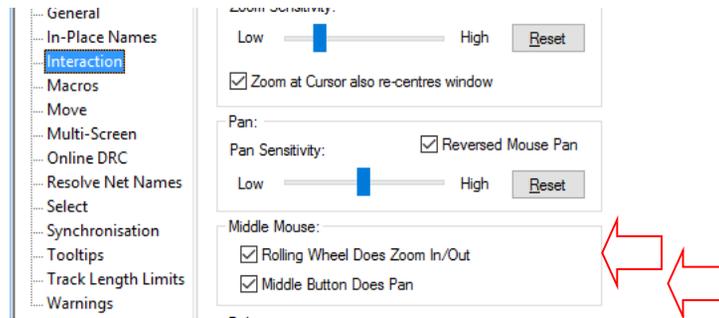


New Commands in Options Dialog

Middle Mouse Wheel/Button Usage Options

On the **Interaction** page of the **Options** dialog, there is a new check box **Rolling Wheel Does Zoom In/Out**.

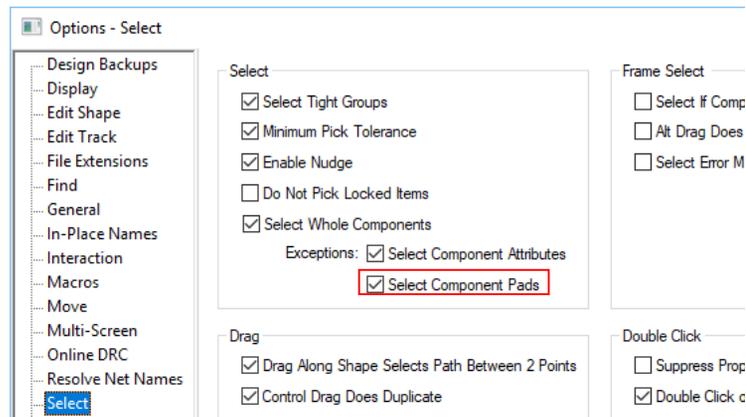
With this option checked (the default state), the existing zoom in/out behaviour will be used. With it unchecked, the wheel does **Pan Vertical** and **Pan Horizontal** if **Shift** is pressed. If **Ctrl** is pressed it will do a **Zoom**.



Also on this dialog is a new option to suppress the **Pan** facility of middle mouse button when the check box is unchecked. The default state is for the check box to be selected to do a Pan using the middle mouse button.

New Component Pads Selection

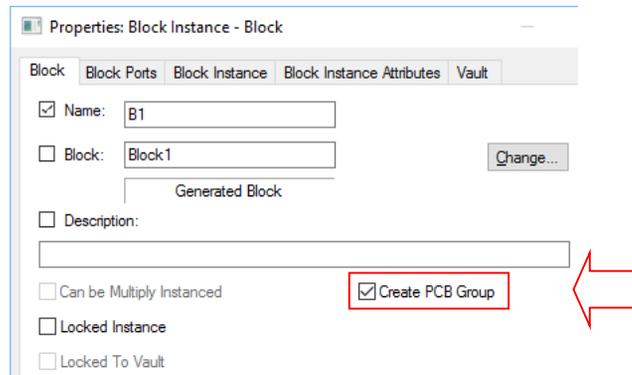
Within the **Options** dialog and **Select** page, there is now an extra option enabled when using **Select Whole Components**. Check the new **Select Component Pads** option to allow selecting of visible Component pads. Uncheck it to have the whole Component selected when picking pads, subsequent **shift-select** on the pad will change the selection to just that pad.



Create PCB Group for SCM Block

You can now mark a **Block** instance (using **Block Properties**) to have a **group** in the PCB.

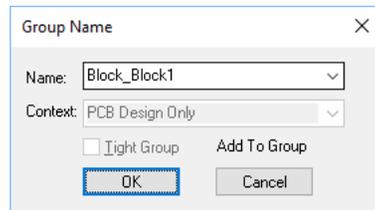
The equivalent group is added to the PCB and **Synchronise Design** will keep them in-line. The group has the same name as the Block, with an optional prefix (which is defined in the PCB, see below).



Block Name Prefix

If required, set the Block Name Prefix for use in the PCB from the **Technology** dialog, **Design Settings – General/Defaults** and **Defaults** button. Choose **Block** and type the name of the **Block Group Name Prefix**:

When viewed as a group, the name will look like this, showing the **Prefix** first, followed by the **Block Name**.



Join Open Shapes

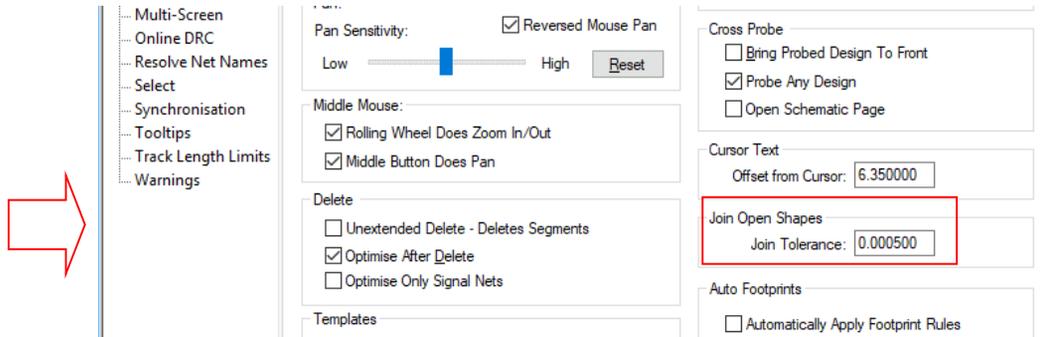
The **Join Open Shapes** option has been added to enable you to select multiple open shapes and then join them into a single shape. This is useful in a number of situations; where shapes have been imported from another system, DXF for example, and because of tolerances of the exported DXF, the shape ends do not meet. The second instance can occur when a design is imported from another system. As with the DXF import, this new feature allows you to join segments which have become dis-connected.

If the shapes to be joined form a complete loop with no crossing lines, the resultant shape will be closed.

This option is available for Doc shapes in Schematic, Symbols and PCB designs. It is also available on Copper in PCB designs.

Join Tolerance Value

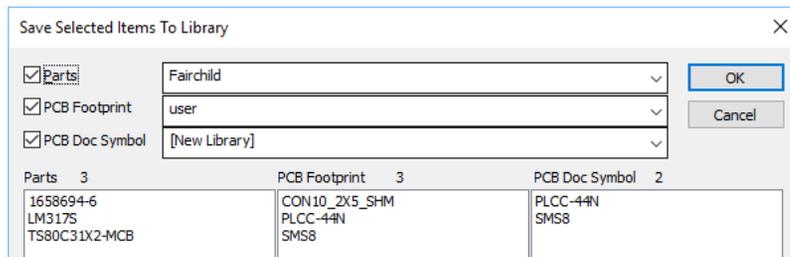
To enable a more accurate joining of shapes and to allow for variations in data, there is a new **Join Tolerance** factor on the **Interactions** page of the **Options** dialog. This is used to determine if the shape ends are close enough to be joined.



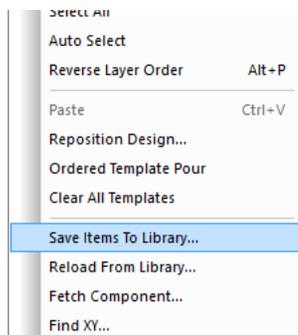
Note: there is already a tolerance built into the importer, in some instances, this is not 'large' enough to cope with the joining of segments. The tolerance is a very small value, you may need to adjust your design Units precision to 5, 6 or lower decimal places to be able to view and set the join tolerance value.

Save Doc Symbols To Appropriate Library

You can now save **Doc Symbols** to the appropriate library when doing **Save To Library** or when dropping an individual design on to the **Library Manager**. This includes **Parts**, **Footprints/SCM Symbols**, **PCB Doc Symbols/SCM Doc Symbols** depending which type of design you are currently editing.

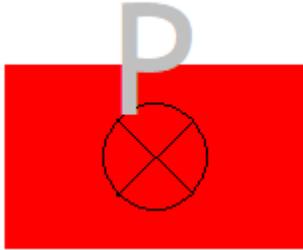


You can also save all library items from a design in one go by using **Save To Library** from the context menu with nothing selected.



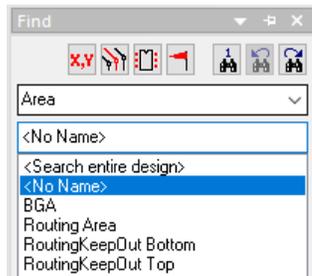
Changes to Error Markers

Error Markers that refer to a single point, such as Single Pin Net or Unfinished Track for example, now have an indicator drawn to show that point. Previously, no specific marker was drawn, this is now an X with a circle around it at the error origin along with the error marker letter.



Find will Find Areas with no name

In PCB, you can now use **Find Area** to find unnamed areas using the new **<No Name>** entry. This will list both unnamed Component Areas and 'free' Areas within the design.



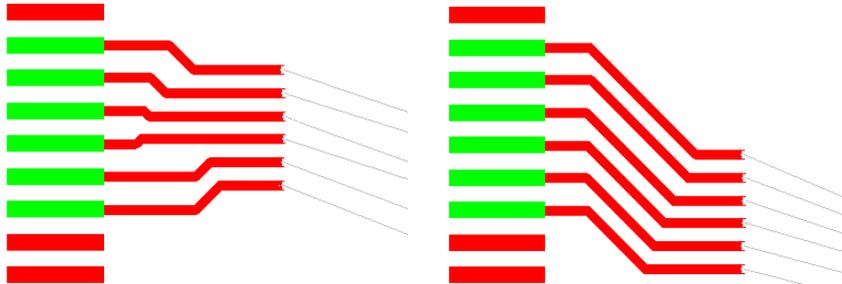
Find Bar Changes to Select All Find Items

Within the **Find Bar**, small change have been made to **Select All Find Items** and **Select Visible Find Items**. Previously, they did not deselect anything before use, now they will deselect all before use unless the **Ctrl** key is held down during use.

Dynamic Starting for PCB Bus Routing

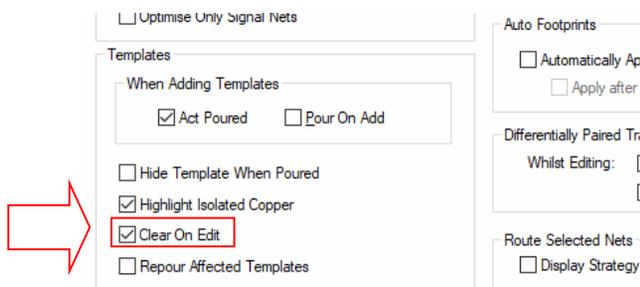
PCB Bus Routing will now add and move tracks from the start items to the moving bus start indicator **on-the-fly**, whereas in the previous release it was only when you clicked to start adding the bus segments.

As the images below show, once started, if the Bus routing start items move down, the start tracks dynamically move to reroute.



New Template Option – Clear On Edit

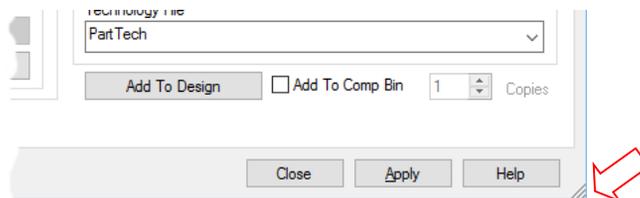
There is a new **Template** option – **Clear On Edit**. This is available on the **Options** dialog and **Interaction** tab. With this enabled, it will remove poured copper when editing the Template.



Library Manager Changes

Resizable Library Manager

All pages within the **Library Manager** can now be resized. As with all other resizable dialogs, use the corner grabber to resize the dialog.

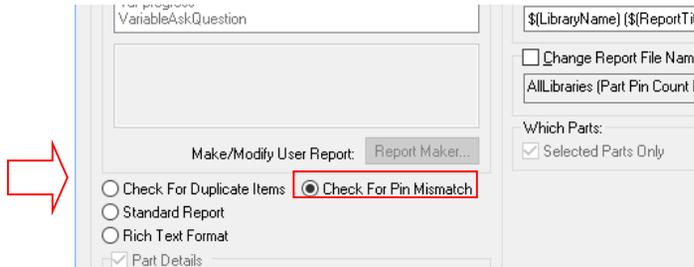


Part Pin Mismatch Report

In the **Library Manager Parts** dialog a new report has been added to **Check For Pin Mismatch**.

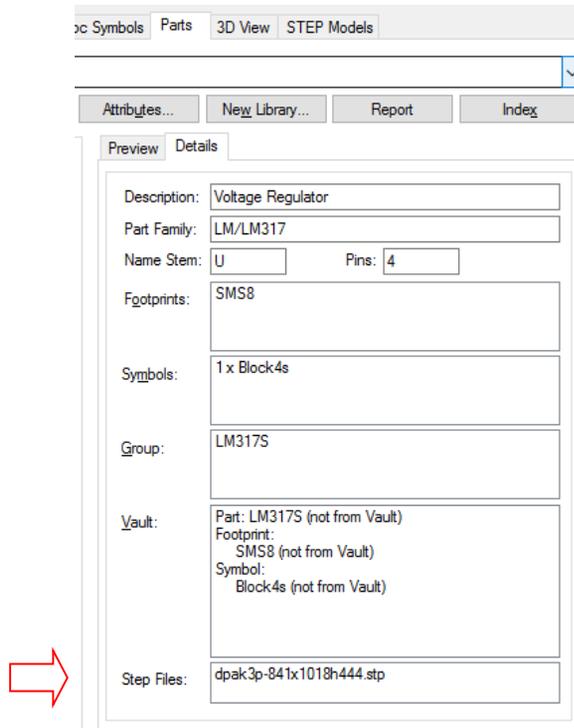
This looks at the expected pin count of a Part and finds and checks against the pin counts on the Parts' Footprints and Schematic Symbols to look for a mismatch. This takes into account Connectors

typically having a single pin symbol for all its Parts. It will also inform you if it simply can't find the Symbol or Footprint the Part references.



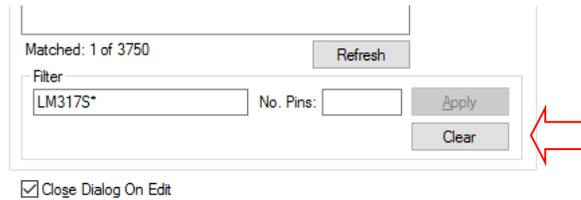
STEP File Names in Part Details

Within the **Library Manager** and **Parts** page, the **Part Details** tab now has a list of **STEP Files** associated with it as defined in the **Part Attributes**. It defaults to the STEP file attribute on a Part, however if it doesn't find one associated with the Part, it will search through the Parts' Footprints and will display the STEP file if it has one associated with it.



Clear Filter

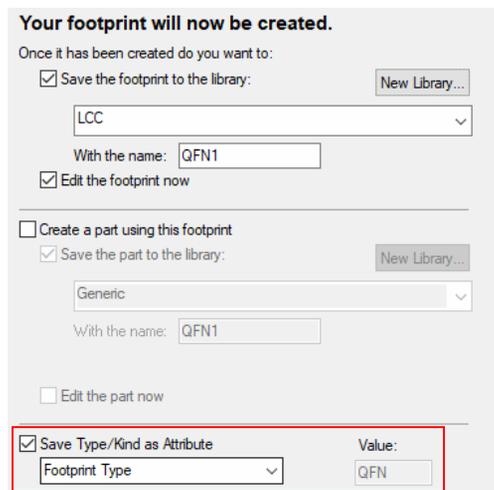
A new button to **Clear** the **Filter** entries in the **Library Manager** has been added. This works the same as clearing the Filter box and/or No Pins box and pressing apply but is much quicker.



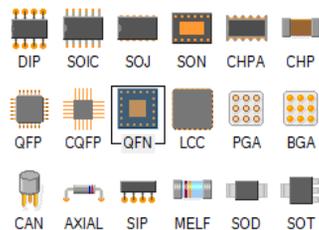
Custom Indexing for Footprints

Footprint Type

The **PCB Footprint Wizard** now gives you the option on the **Finish** page to attach the 'type' of footprint as an attribute to the footprint.

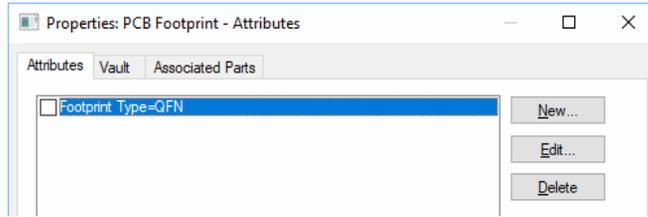


The footprint 'Type' or 'kind' is taken from the **PCB Footprint Wizard Type** page using the Type selected for creation:



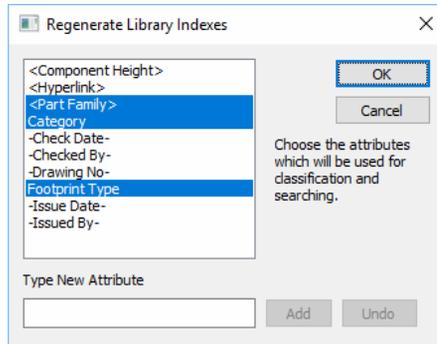
When the Footprint created is subsequently edited, the **Footprint Type** attribute will be available in the **Design Properties** along with its value. This will be an editable value if you need to update its

Type. Likewise, if you wish to categorise any of your existing Footprint library footprints, this Attribute and Value can be added using the **Attribute Editor**.



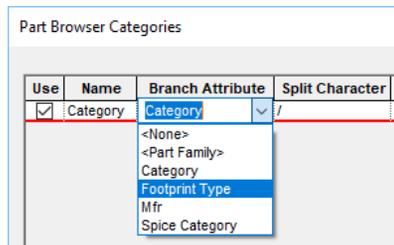
Custom Footprint Indexes

You can now Index more than just the pin count. This is now similar to the Parts indexing where you can index any Attribute name available or a custom Attribute when typed. When the **Index** button is selected on the **PCB Footprints** page, this dialog is now displayed:



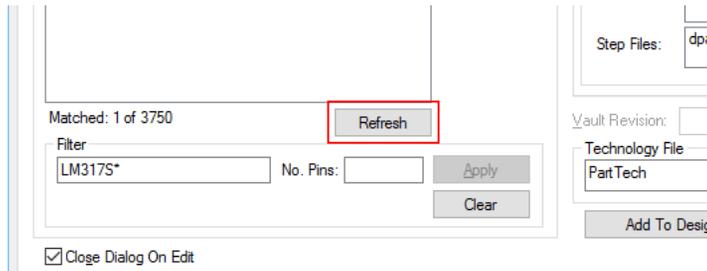
Using the Footprint Type

With the Footprint Type now in the footprint definition, you can now use it in the Parts index. When indexing your Parts, chose Footprint Type or type Footprint Type and then re-Index the Parts. Once in the Parts index, it can be used for categorisation within the **Parts Browser** for example.



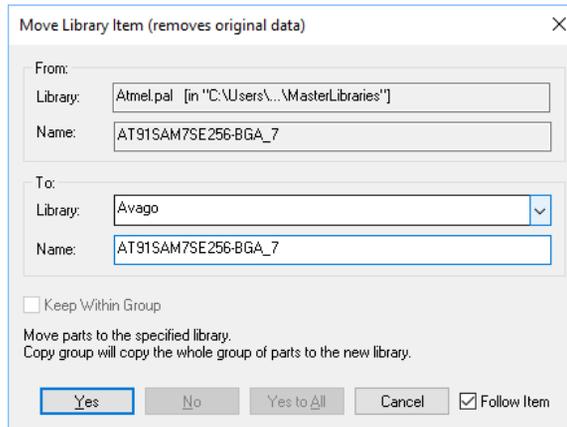
Manual list Refresh in Library Manager

A new **Refresh** button has been added to all pages of the **Library Manager** and including the **Folders** page. This allows you to manually refresh the contents lists. This would be used when manually adjusting the library environment that the Library Manager does automatically know about. For example, if you manually remove a library file that has previously been accessed because it is within the library search path. In this instance, the refresh option will re-search the library path.



'Follow Item' on a Part option

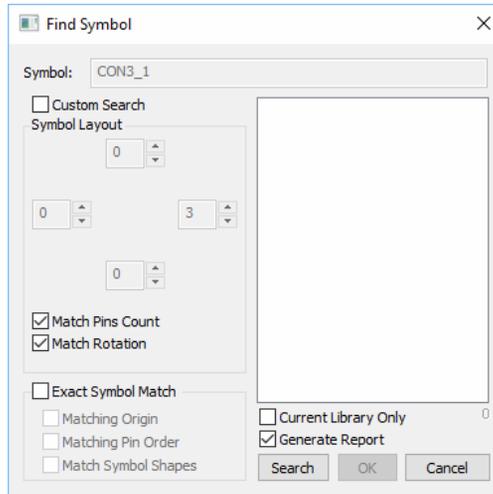
The ability to follow a Part to a new library has been added. With the **Follow Item** check box selected on the **Move** or **Copy Library Item** dialog, this will action the option, then when copying or moving the Part will open the Part on the new library and select it in the **Contents** list.



Find Symbol

The **Find Symbol** option has been added **Schematic Symbol Library** page. This new option allows searching of symbols that fit certain criteria or match a preselected criteria using a symbol referenced from the list.

This dialog can work in two modes, the first is from a pre-selected **Symbol** name off the symbol contents list on the Schematic Symbol Library page, the second is as a **Custom Search** using defined criteria.



The **Find Symbol** option can take parameters passed through on specific, **Symbol:** name selected from your library. As the name is pre-selected from the Contents list in your library, it is not editable here. Parameters are displayed on this dialog, extracted from the symbol, such as Pins on the Symbol Layout.

Alternatively, by selecting the **Custom Search** check box, you can define your own parameters.

The Find Symbol tool will generate a value for the Part you have selected in the Library Manager page, these values will represent the pin layout in relation to the Symbol selected. The values represent the number of pins and in which direction they are found in relation to the Symbol.

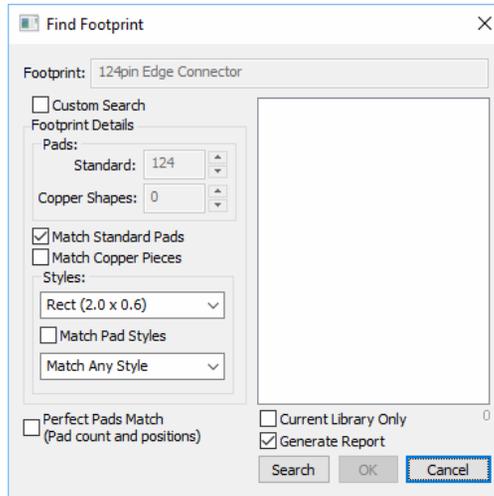
At this point you can perform two types of searches all with different results based on the search options and criteria. By default the tool will run a simple 'pin match' basic search.

Check boxes on the dialog allow various parameters to be defined, these are used to add extra filtering the search.

Find Footprint

A new **Find** tool has been added to the **PCB Footprint Library Manager** page. This allows searching of existing Footprint libraries and reports matched items.

Footprints that fit defined criteria or match a preselected **Footprint** name from the list.



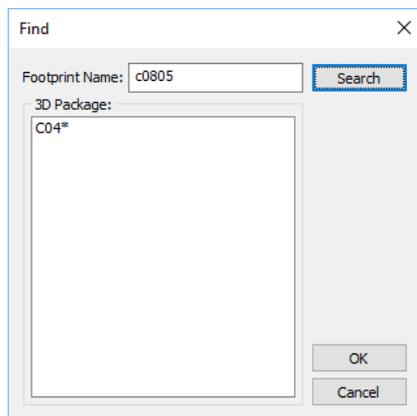
You can make a search for a particular footprint in open libraries by pressing the **Find** button from within the **PCB Footprints** tab of the Library Manager with a footprint selected. Or you can search for a user defined number of pads and copper shapes by entering the find tool with no selection or by clicking the * button next to the name of the footprint.

The Find Footprint tool will generate pad values for the footprint you have selected in the Library Manager page, these values will represent the pad count, copper shape count and styles present in the Footprint selected.

At this point you can perform different types of searches all with different results based on the search options and criteria.

Find Footprint in 3D Packages

Within the **Library Manager** under **3D View** page, a new **Find** button has been added. This enables you to search for Footprints that are referenced by the 3D Packages.



Rename in Library Manager – Replace Symbol References in Parts

In the **Library Manager** under the **Rename** button for **SCM Symbols** and **PCB Footprints**, there is a new check box for the **Replace References** feature. This will find and then enables you to replace all references of the old symbol name with the new symbol name in your open Part Libraries.

Swap Symbols Option

Within the **Parts Editor** page of the **Library Manager** there is a new **Swap Symbol** button.

This enables you to find and replace all references to an old symbol name or multiple symbol names and to replace it with a new name in your Parts library.

It also gives you the option to add an alternate footprint to Parts that contain an existing one (e.g. parts containing footprint x can have y added to their footprint list as an alternative).

Optionally choose to add the swapped Footprint or Symbol as an **Alternate** symbol in the Parts Library. Selecting this option will force it to check the footprints you are adding using a pin count match before adding them as alternatives in the found Parts.

Select the **This Library Only** option to only make swaps in the current selected Parts Library. If **All Libraries** has been selected in the Parts Library dialog, this check box will be greyed out as not available. With a single library selected in the Parts dialog, the search will search and swap (where applicable) in all libraries unless this option is selected.

Select the **Delete Old Symbols When Done** option to delete the old symbols once they have been found and replaced in the Parts library.

Select the **Generate Report** option to report any changes and to report the success of name replacements when the option is run.

Search

On selection of the **Search** button, when the search is complete, you will be presented with a list of Parts that use the Footprint or Symbol and the relevant Parts libraries. From the list, you can choose which items to replace before continuing. The search will be made on an entered Footprint or Symbol name defined as **Old Name** or by using the results of the **Load Report** option.

The **OK** button will start the process and replace all references to the previous Footprint or Symbol with the new names. If the new name list has more items than the old name list you have entered, the Swap Symbol option will replace all Parts with the single new Part name. This enables you to swap out the symbols to multiple old Parts with a single new Part.

Load Report

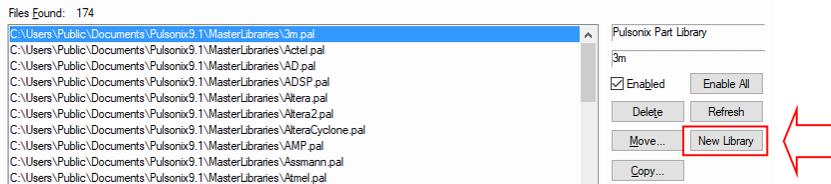
You can enter a comma separated list of Parts to use as your search and a comma separated list of parts to replace them with.

You can also use the **Load Report** button to open a '**Similar Symbols to (X)**' report produced through the **Find Symbol** option on the PCB Footprints or Schematic Symbols tabs of the Library Manager. This will use the perfect match option to fill the old names list for you. When you have entered the Parts to search for, click the **Search** button. A progress will show the progress of the search. The search is **NOT** case sensitive, however it requires the Part name to be exact, including spaces, otherwise the name will not be found.

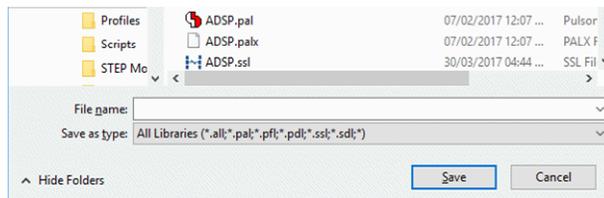
Be careful when using this tool as no check is made to see if the replacement Footprint or Symbol is actually available, unless using the option with footprints to add alternative footprints.

Create New Libraries directly from the Library Manager Folder tab

There is an additional button on the **Library Manager Folders** dialog. The **New Library** button allows you to create all five library types at once with the same name (SCM Sym, SCM Doc Sym, Parts, PCB Footprints and PCB Doc Sym). These empty files can then be populated as required using the various methods.



All five library types are saved at the same time using the **Save As** dialog:



Part Description Using Substitute Attributes

You can now create a ‘dynamic’ Description field for a Part using Substitute Attributes. This can be done in the **Library Manager** when editing a **Part** (using New or Edit).

The **Description** field can be created from Attributes that appear in the Part definition. The format of the name will be:

Syntax: **Res_Value (%%Value%%)** with the resultant value displayed as: **Res_Value (10k)**

The Parts Attributes field would look like this:

Attributes		
Part Name	Auto Name Parts	4308R-102
	Category	Resistor/Resistor Pack
	Value	10K
	MFR	Tyco

The **Parts Details** page will show an edited **Description** field like this:

Part Name: 4308R-102 1K

Description: Res_Value (%%Value%%)

Part Family: 4308R/4308R-102

Name Stem: R FPGA

Pin Count: 8

Footprint: SIP8

When the Part is used in the design, the Component **Description** field is populated with substituted values.

Variant Id:

Position: 2710.00 5740.00 Mirrored Allow Automatic Changes: Auto Footprints

Angle: 0.0

Layer: <Top Side>

Part: 4308R-102 10K

Description: Res_Value (10K)

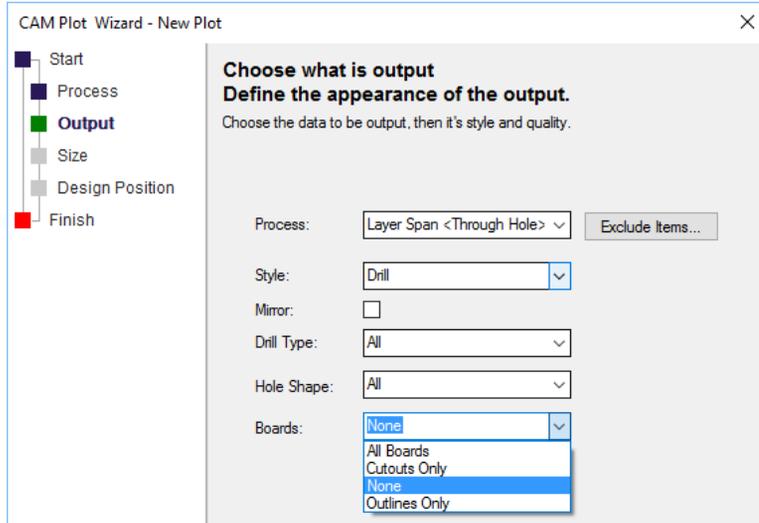
Part Family: 4308R/4308R-102

Footprint: SIP8

CAM Plots – Drill Output available for Board Outlines & Cutouts

In the **CAM Plots** option, for a **Drill** output within the **CAM Plot Wizard**, you can now add board outlines and area board cutouts to the drill output.

On the **Output** page, there is a new option to select **Boards**: From here, you can choose **None**, **All Boards**, **Cutouts Only** and **Outlines Only**.



All Boards - means all board outlines including **Area Board Cutouts**,

Outlines Only - does not include **Area Board Cutouts** or **Cutouts Only** (only **Area Board Cutouts**). Area Board Cutouts can be plated or non-plated, and these are selected using the **Drill Type** above.

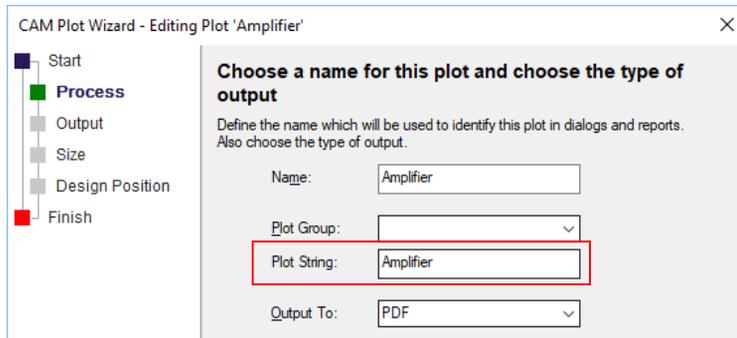
Select **None** to prevent any Board outlines being output.

CAM Plot – Plot String Attribute

You can optionally provide a Plot String on a CAM Plot output. This is used to satisfy any Attribute Positions of type <Plot String> placed on your design. This could be used to define a reference number or other identifier on your output.

When adding an attribute position to the design, if you select <Plot String>, the value cannot be added here but is instead, added when the **Plot String** is satisfied in the CAM Plot dialog.

To define the Plot String, it can be added using the **CAM Plot Wizard** or directly in the **CAM Plot** dialog.

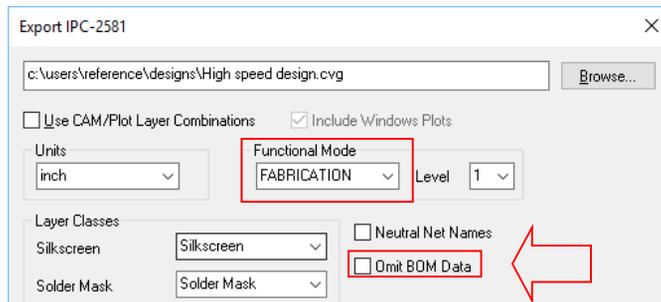


Name	Enabled	Device	Process	Scale	Rotate	Mirror	Position	Area	Step Repeat	Exclusions	Plot String
Amplifier	<input checked="" type="checkbox"/>	PDF	Amplifier	Auto Fit	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>	<input type="checkbox"/>	Y	Amplifier
PSU	<input checked="" type="checkbox"/>	PDF	PSU	Auto Fit	Auto Rotate	<input type="checkbox"/>	Auto Shift	<Design Extents>	<input type="checkbox"/>	Y	PSU

Whatever value you type in the Plot String box will be added to the plot once output. This is plotted at the <Plot String> attribute position.

Export IPC-2581 - Omit BOM Data

When exporting the IPC-2581 netlist, there is now a new check box, **Omit BOM Data**. This is only displayed when choosing a **Fabrication** output which normally would contain BOM data but enables it to be removed for security reasons and protection of design IP.



This feature has also been back fitted to Pulsonix 9.1

Toggle True Width For Layers

Alan 01/06/18 - Yes

Added a command to toggle true width for all layers. This is available on the context menu of the Layers Bar.

Report Maker Changes

Access Other Tables in PDC Database

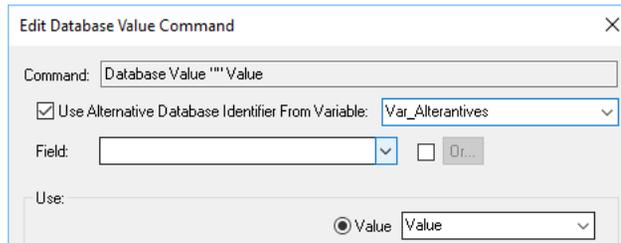
Report Maker – Database Value

Within the **Report Maker** option and the **List of Components** command, the **Database Value** command now has an extra ability. As well as reporting the database value from the current Component you can now use it to report a value from a related Component using the required Component's key value identifier.

This is done by specifying a variable name that at run time will contain the key value identifier for the alternative component. For example this identifier could be taken from an attribute on the Component being reported.

Use Alternative Database Identifier From Variable

Check the **Use Alternative Database Identifier From Variable** check box and then provide the **Variable** name from the drop down list.



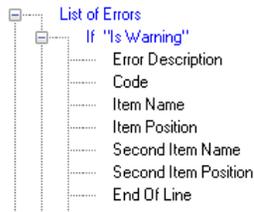
The screenshot shows a dialog box titled "Edit Database Value Command". It contains a "Command" field with the text "Database Value "" Value". Below this is a checked checkbox labeled "Use Alternative Database Identifier From Variable:" followed by a dropdown menu showing "Var_Alterantives". There is also a "Field:" label with an empty dropdown menu and a button labeled "Or...". At the bottom, there is a "Use:" section with a radio button selected for "Value" and a dropdown menu showing "Value".

This command has also been back fitted to Pulsonix 9.1

Adjacent Nets in Report Maker – List of Errors

Report maker can detect the **Adjacent Nets** errors in a **List of Errors** command by using the **Is Warning** command. You can report the net names using the new **Item Name** and **Second Item Name** commands.

This might be used like the example below:



This would report a message like this:

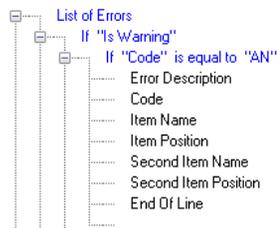
```
Adjacent Nets (AN) Between (57.800 22.496) and (43.800 31.196). Layer 'Top'. Actual 16.483
```

```
Code: AN
```

```
Net1: PG0 (57.800,22.496)
```

```
Net2: PE5 (43.800,31.196)
```

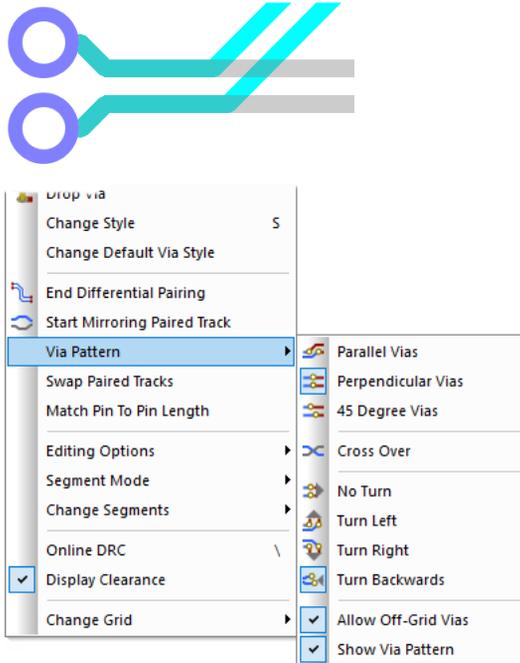
Another example to report adjacent net rule errors might be like this:



High Speed Option Changes

Differential Pairs - Turn Backwards direction on Via Pattern

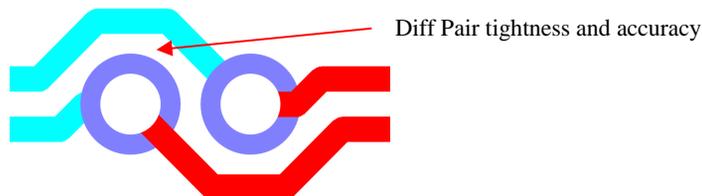
When swapping layers with Differential Pairs, the **Via Pattern** selectable can now be **Turn Backwards**. This means, once the layer swap has been made, the pair will turn back on itself.



Improved Accuracy of Differential Pairs

Improved tightness of Differential Pairs on a Layer Swap

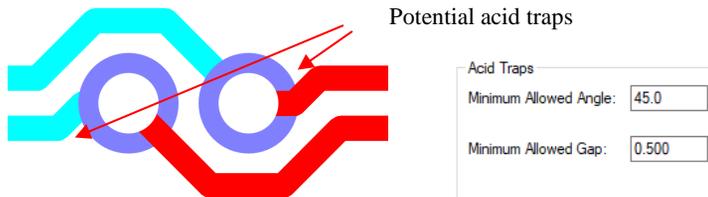
During Differential Pair editing, when applying a **Layer Swap**, a predefined pattern is added. In version 10, the accuracy and 'tightness' of the shape of the tracks around the via pattern when using a **Parallel** style has been significantly improved.



Acid trap removal using Minimum Allowed Gap Spacing Rule

During Differential Pair routing, when performing a **Layer Swap** using the **Parallel** pattern, the Acid Trap rule defined will be used. Using a combination of the **Minimum Allowed Angle** and the **Minimum Allowed Gap**, you can control the removal of acid traps.

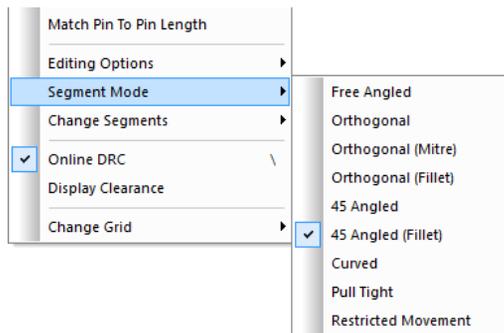
These values are defined in the **Technology** dialog under **Spacings, Design Level** and the **Track** tab. Under the **Acid Traps** rule, you can change the values of the **Minimum Allowed Angle** and **Minimum Allowed Gap** to that required.



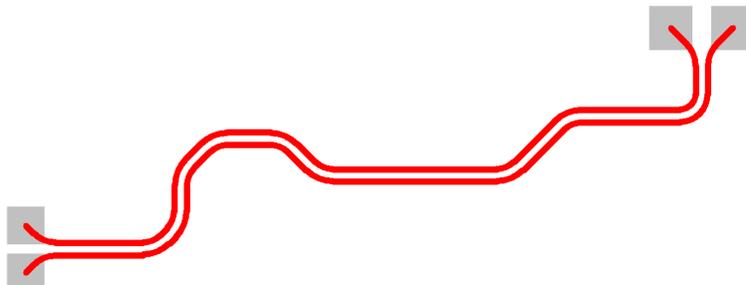
To utilize this fully, an angle entered for the **Minimum Allowed Angle** will control the acid trap removal to ensure that the track entry to the via will conform to this rule. If you require additional control, you can specify the **Minimum Allowed Gap** also. This will mean that if the angle is violated, the track will then be backed off by the gap defined. If the gap is set to zero, the minimum required to remove the violation will be applied.

Improved Differential Pair Routing – Fillet Mode

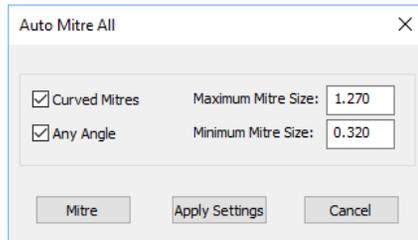
While routing Differential Pairs, you can now switch **Segment Mode** to use one of the two new additional routing modes: **Orthogonal (Fillet)** and **45 Angled (Fillet)**.



When using 45 Angled (Fillet), you can produce Differential Pair tracks with smooth filleted corners.

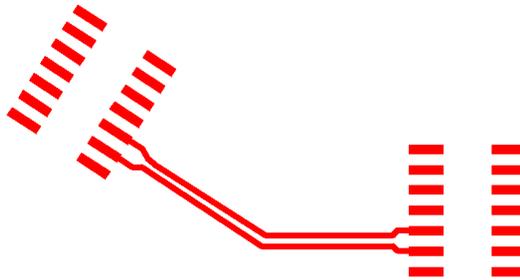


Fillets can also be applied to Differential Pairs on existing tracks that were routed previously by selecting the corner and using Edit Mitre from the context menu. The Mitre used (45 degree or Fillet) will depend on the setting using from the Tools menu, Auto Mitre, Auto Mitre All. Both Curved Mitres and Any Angle should be enabled.

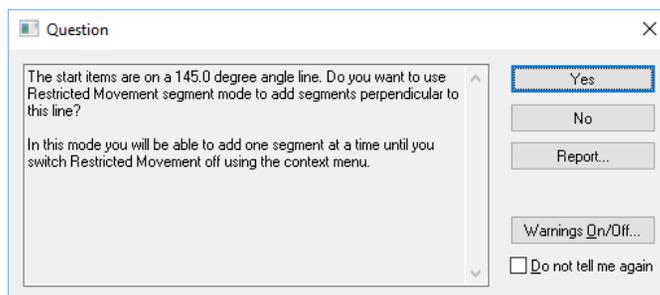


Improved Differential Pair Routing – Mirror Diff Pair routing from Angled Components

Differential Pairs can now be routed out of pads at any angle.



Do this by start mirroring a Differential Pair track from two pads that are at an odd angle (not a multiple of 45 degrees), you will be presented with a warning dialog asking if you want to enter **Restrictive Movement** mode to add segments that are perpendicular to the line through the start pads.



Choose **Yes** to do this. The restrictive movement angle will be automatically set and the track segments will be at the correct angle. If this is an operation you do often on non-45 degree components, then you can switch this message off using the check box on the dialog.

Using this mode

After cornering, the mode stays and you can turn in 45 degree steps relative to the start angle. When close together and you click, it will enter adding paired tracks still in the restrictive mode. Add a

74 Pulsonix Version 10.0 Update Notes

segment or two and then use the context menu to switch **Restrictive Movement** off and now you can add the segments in your normal segment mode.

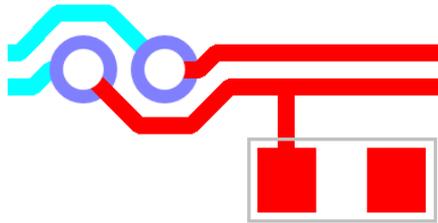
Choose **No** to the dialog to add the mirrored segments in a direction that is a multiple of 45 degrees closest to where you clicked to start mirroring. (This is how Version 9.1 and previous versions worked).

Note: **Restrictive Movement** mode always allows you to turn a multiple of 45 degrees now. In 9.1 this only worked if you set the angle to zero.

Finish Track On Differential Paired Track (Spur)

You can now finish a single track on a Differential Paired track segment provided the track is not an arc and is on the same net. To allow this happen, in the **Technology** under **Nets – Differential Pair**, the pair must have the **Allow Spurs** rule enabled.

With the single track over the Differential Paired track to connect to, on drop, it will unpair a small section of the segment and join the track to it. If you also have **Optimise After Edit** enabled in **Options, Edit Track**, the connection from the single track will be optimised automatically.



New Serpentine Skew Modes on Differential Pairs

Selection of single track on a differential Pair

A change to the way in which **Serpentine** operates on **Differential Pair** routing has been made. When a single track segment of a Differential Pair is selected, when using the **Dynamic Serpentine** mode, regardless of the position of the cursor relative to the 'other' Differential Pair track, the Serpentine will now only be applied to the selected segment. Previously, the position of the cursor when in this mode dictated which track was selected from either pair.

Serpentine Skew Modes on Differential Pairs

Serpentine mode has been improved for Differential Paired tracks. There is now biasing, selectable from the context menu, to enable you to bias the track upwards so that the bottom track is not affected, to centre the bias around both tracks of the pair and to bias the serpentine track to the bottom and to have the bottom paired track move to allow space for routing.



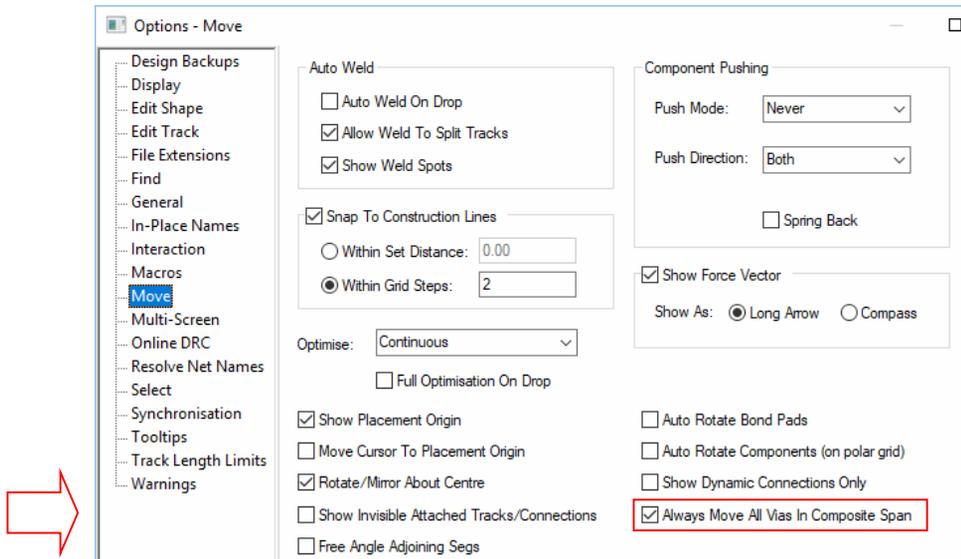
Biased Top

Biased Both

Biased Bottom

Move All Vias in Composite Span

When using the **High Speed** option and **Composite Vias**, there is a new option in **Options** under **Move** to **Always Move All Vias In Composite Spans**. This is only applied when a single via in a span is selected and move performed. This now defaults to a checked status to always be on so that the Composite Via moves as one unit. Uncheck it to get behaviour like version 9.1 where just the single selected via moves.



Network License Server (NLS) Changes

A complete rework of client-server interaction for network licensing has been undertaken for Pulsonix 10.

On the server side:

NLSD new application that does the actual licensing runs as service or application
 NLSM manager application that controls NLSD either on server or from another computer
 NLSL log viewer application used from NLSM as an easy way to view log files

Client side:

No visible change for clients, with all changes being within the client application itself. The separate **PulsonixMonitor** application that used to run as a hidden program to maintain the heartbeat signals to the server is now obsolete. A separate thread inside the main Pulsonix application now manages heartbeats. Performance is better from the client side, allowing quicker application startup time.

The separate **Network Licensing User Guide** has full details about installation and management of NLS.

Default Supplied Technology Files Changes

All Schematic and PCB Technology files have been set to the **Basic Set** for **Style Naming Rules** for **Hatch, Line, Pin, Pad, Text, Connection** and **Track**.

All PCB Technology files have the 'Footprint Type' attribute added as Any Item, PCB Design Only.

Unattended Installation

The Pulsonix 10.0 installer has been modified to allow IT Managers of large networked installations, the ability to mirror an installation with ease across other clients. This uses the standard Microsoft network installation where an install set is pushed onto clients.

An additional user's guide named Pulsonix Unattended Installation.pdf is provided on the Pulsonix web site under documentation.