



Pulsonix Design System V10.5 Update Notes

2 Pulsonix Version 10.5 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2019

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: 02/08/19 iss 2

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 10.5 UPDATE SUPPLEMENT	6
Installing the New Version of Pulsonix	6
Licensing	6
Version 10.5 Update Notes	6
Installation	6
NEW IN VERSION 10.0 - NOT PREVIOUSLY DOCUMENTED.....	7
Drag and Drop facility for Library Manager	7
Design Colours Loaded During Eagle Import	7
DXF Output in Panel Editor	8
New Standard Report added - Tracks by Layer	8
Graphics Declutter Option	9
Layer Stack Items	10
NEW IN VERSION 10.5.....	11
File Open ‘View Only’	11
View-only PCB and SCM Designs	11
New ‘Detach from Vault’ option	11
Update Vault Cross-References.....	12
Database Connection Option – Database Type.....	13
Area Attribute Rules in Schematics	14
Plot Areas in Doc Symbols	16
Component Place Rules in PCB	16
Hierarchical Block Instances.....	17
Regenerate Block Instance – Reset Pins.....	17
Delete Block Port.....	17
Block Instance – Edit Symbol in Place	17
Text Callouts with Solid Colour Background.....	18
New Doc Symbol Type - Bus Reference.....	19
Orcad Netlist Export in SCM – Suppress Net/Part Names	20
Selection of a Signal Reference with a Single Pin	20
Electrical Rules Check - Show Errors Bar	21
Auto Weld Component To Bus in Schematic Designs	21
Multi Connect in Schematic Designs	23
New Default Settings – Connections Gap	25
Synchronise with Testpoints Changes.....	26
Design Settings switch for Synchronisation of Net Class Changes.....	26
Naming – Fill-In Gaps In Number Ranges	27
Change Part – Default Selected Library	27
STEP 3D Preview Enhancements	27
STEP Model Folders.....	27
Spotlight Position in STEP 3D Preview	27
Include Enclosures in 3D View switch in STEP 3D Preview	28
New Save Enclosure Positional Changes option in STEP 3D Preview.....	28
Exploded View in STEP 3D Preview.....	29
Layers Page in STEP 3D Preview.....	30
Board Transparency option in STEP 3D Preview	30

4 Contents

New Measure feature in STEP 3D Preview	31
Clash Markers in STEP 3D Preview	32
Support for 3D Space Mouse in STEP 3D Preview	33
Changes to the Customise Dialog	33
Star Points for other Pad Types in PCB	33
New Items in Find Dialog	34
Track Impedance Tooltip Calculations	35
Set Toggle Layers Setup	35
Set Pour Order.....	38
Set Pour Order now a Modeless Dialog	38
Design can now be Dimmable during Set Pour Order.....	38
Drill Drawing Layer & Idents	39
Drill Drawing Layer.....	39
Display of Drill ID using Attribute Position	40
Coloured Drill Sizes & Drill Table	41
Case Sensitive Drill Size Idents	41
Panel Editor - Tab Rout Changes.....	41
Changes to Technology Files	43
Merge Styles in Technology Dialog.....	43
Net Names Page – Delete Multiple Rows.....	44
Pour Squares Off Pad Corners	45
Square-Ended Thermal Spokes.....	46
Use Centreline of Template for Pouring	46
Ability to Enable/Disable Rules.....	47
Selection of Disabled Rules in CSV Export	47
Ability to add Notes to a Rule	47
Cell Notification of Rule Errors.....	48
Grid Column Filters on Net Names Page	49
Suppress Unconnected Lands for Micro-Vias	50
Display Implied Junctions moved into Technology	50
Changes to the Grids Dialog.....	50
Display Implied Junctions on Individual Pins on a Symbol	51
Frame Selection of Construction Lines	51
Insert SMD Pad in Footprint Editor.....	51
Properties	52
Component Attributes	52
Via Attached Tracks & Templates/Planes	52
Error Marker Information.....	52
Lock Aspect Ratio and Scale for a Bitmap	53
Lock Aspect Ratio When Dragging a Bitmap Edge	53
Attachment Information for Dimension & Text Callout.....	53
Layer Stack Appearance.....	54
Edit Radius/Diameter in Properties	55
Next Available Name in Component Properties	55
Measure Tool - Angle tab	55
Double Click to Edit Mitre changes.....	56
Changes to CAM Plot Dialog.....	56
Ability to Switch Off CAM Plot Reports	56
Additional Outputs Types Available Through CAM Plot Dialog	56
Named Output Devices in CAM Plot Dialog.....	57
Separate Options in Reload From Library for Styles and Positions	59
Recheck Locked Errors.....	59
Create Sub-Net Highlighting and Dimming	60
Change Variant.....	60

Add Copper & Template To Net During Insert61
Intelligent Gerber Import Drag and Drop Files.....61
Report Maker Changes62
Editing Differential Pairs over Area Boundaries69
Changes to Default Supplied Files.....69

Version 10.5 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 10.5 on top of your existing V10.0 installation. If upgrading from V10.0, 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 10.5 requires a new license if you are a new user or upgrading from any older version of Pulsonix earlier than V10.0. If already using V10.0, you will not require a new license. The new license 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 in the same location as the current one but make a backup copy first or rename it. 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 10.5 Update Notes

These Version 10.5 Update Notes are available in the Pulsonix installation folder under \documents. Alternatively, select the direct link within the product itself: **Help** menu | **Online Manuals> Pulsonix V10.5 Updates Notes**

All previous Update Notes are available on the Pulsonix web site under **Documentation**.

Installation

The installation has been changed so that all Pulsonix 'documents' (Master Libraries, Technology files etc.) are now located under user\documents\Pulsonix10.5 rather than being placed in public documents\PulsonixXX

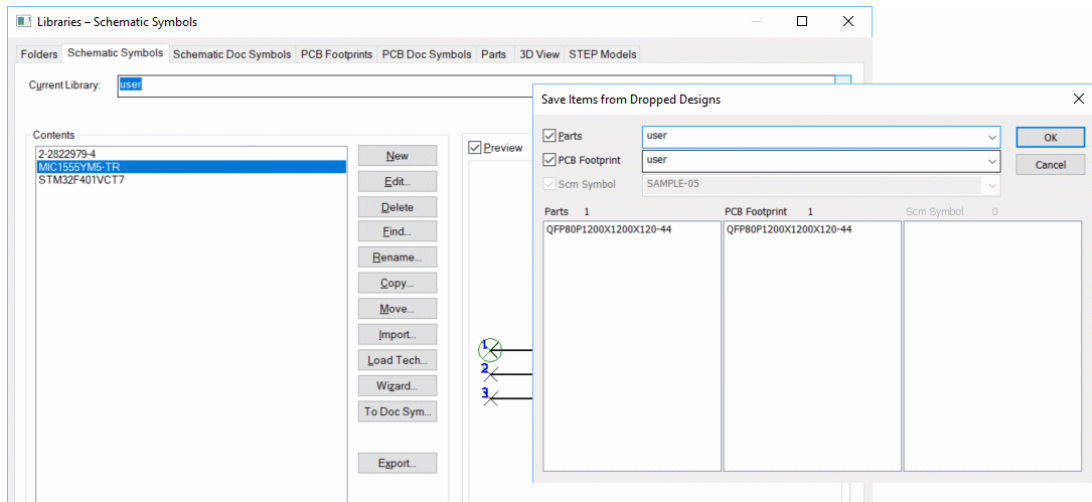
New In Version 10.0 - Not Previously Documented

The feature(s) below appeared in Version 10.0 but were not documented. They have been added here for your reference:

Drag and Drop facility for Library Manager

It is now possible to drag and drop other EDA libraries in the supported ASCII formats, such as OrCAD .edf library files, onto the Library Manager where it will use the 'drag and drop files' mechanism. This has also been expended for all EDA library formats supported.

To summarise this option, when a file is dropped onto the **Library Manager** it first displays the Technology file chooser dialog for each of the library types, then displays the **Save Items from Dropped Designs** dialog. From here, you can choose the library to save the library item into and choose the type and items to save. Pressing **OK** starts the import procedure.

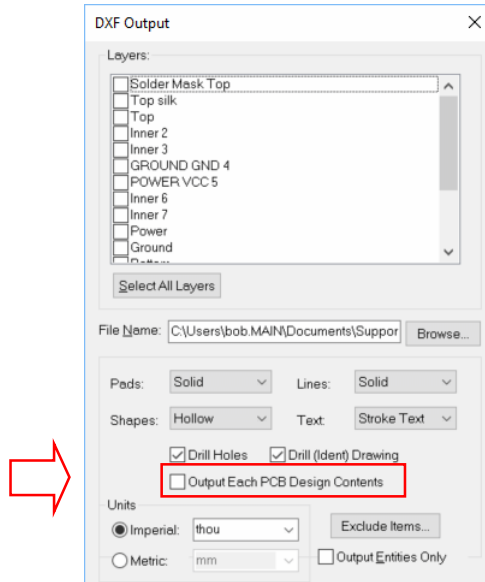


Design Colours Loaded During Eagle Import

When importing designs from Eagle, the colours within the original designs are also imported to preserve its original look. These are imported for both Schematic and PCB designs.

DXF Output in Panel Editor

From within the **Panel Editor**, you can now export **DXF** files from the **Output** menu.



This is the same as the normal **PCB Design DXF Output** except there is a new check box on the DXF Output dialog called **Output Each PCB Design Contents**, for including PCB instance contents.

When output, extra DXF layers are created in the DXF file for each of the three panel shape types used in the panel (Panel Outline, Panel Tab Route and Panel V Score). If the shape doesn't exist in the Panel design, that layer won't be exported.

Note, if reading this with a view to using it in the version 10.0 product, you will also need to activate the menu option using the **Reset Panel Toolbar** in the **Customise** dialog under the **Toolbars** tab.

New Standard Report added - Tracks by Layer

A new report has been added to the standard reports dialog and is named **Tracks by Layer Report**. This reports the layer and the Track Widths along with the Track Style Name used on that layer.

A sample report might look like this:

Tracks by Layer Report

```
Report Written : 10/01/2019 02:47:13 PM
Design Path    : C:\Documents\Designs\16995\NSP8744A.pcb
Design Title   : USB Interface Module
Created        : 19/12/2018 11:20:32 AM
Last Saved     : 19/12/2018 11:28:01 AM
Editing Time   : 58 min
```


Units : mm

Layer No. : 1

Layer Name : TOP

Track Widths	Track Style Names
15.000	Track Style 15
60.000	Track Style 60

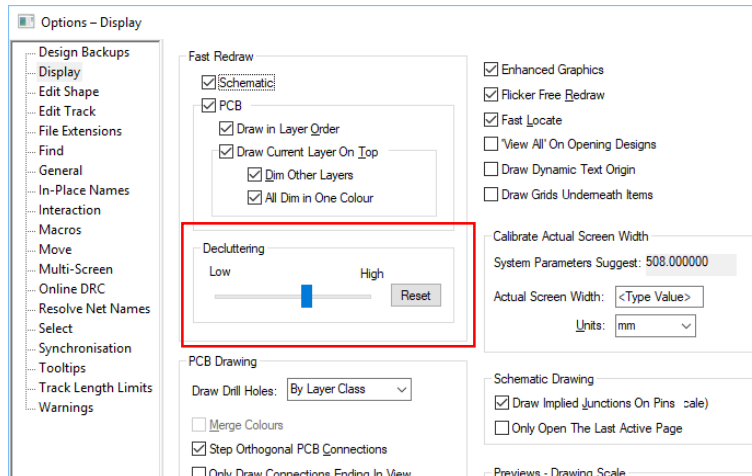
Layer No. : 2

Layer Name : BOTTOM

Track Widths	Track Style Names
15.000	Track Style 15
30.000	Track Style 30
50.000	Track Style 50
60.000	Track Style 60

Graphics Declutter Option

From within the **Options** dialog on the **Display** page is a new **Declutter** feature. By removing insignificant items using declutter, the screen redraw will be increased. This is utilised in the PCB design editor only and has no impact on Schematic designs.



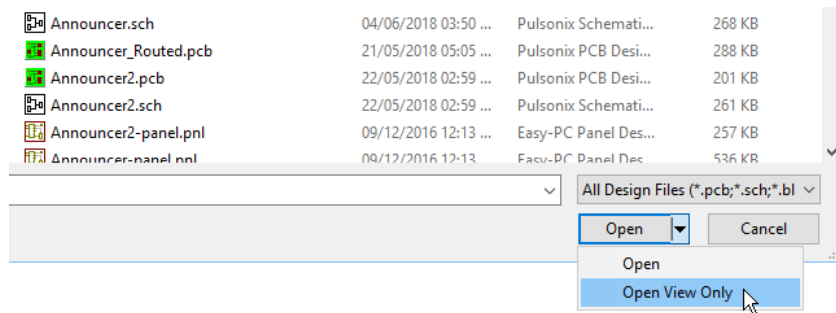
This works by varying the amount of detail, for certain design items, shown at different zoom levels. When items become too insignificant to be meaningful at lower zoom levels, they are not drawn. The degree of declutter (point at which items are no longer drawn) can be varied by using the slider.

New In Version 10.5

The following documentation details features specifically for Version 10.5

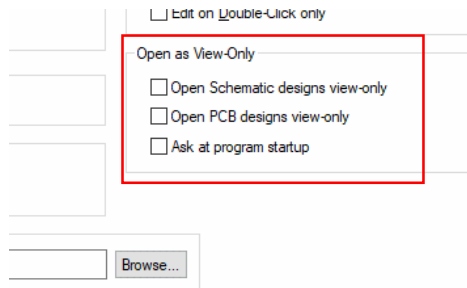
File Open ‘View Only’

On the **File Open** dialog, the Open button now has a drop-down button that provides you with an extra option **Open View Only**.



View-only PCB and SCM Designs

There are new settings that allow you to specify whether to open PCB or SCM designs as view-only. These are available in the **Options** dialog and **General** page. These settings are only enabled if you have none of that type of design open. You can also choose to be prompted for these settings each time the application is started.



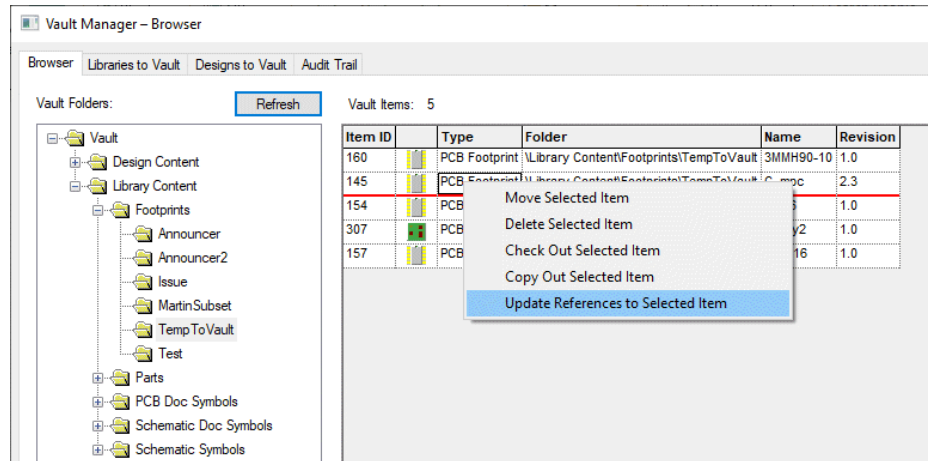
New ‘Detach from Vault’ option

There is a new option available from the **File** menu called **Detach From Vault**. This removes Vault information from the current design or library item so it may be added back into the vault as a new item rather than as a new version of the original item.

Update Vault Cross-References

It is now possible to access the **Update Vault Cross-References** dialog from the **Vault Browser** in addition to its being displayed as part of the checking-in process.

It is available as the **Update References to Selected Item** option on the **Vault Browser's** items list context menu.



This provides you with access to the **Update Vault Cross-References** dialog.

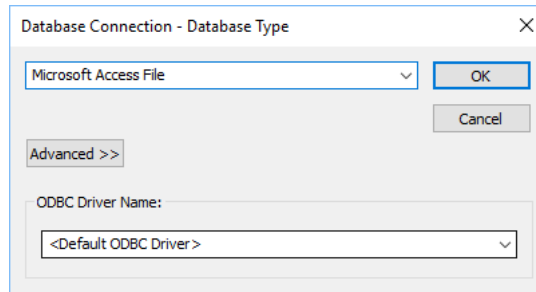
The screenshot shows the 'Update Vault Cross-References' dialog box. It contains a table with columns for Item ID, Type, Name, Revision, Used by ID, Type, Name, Revision, and Update?. The 'Update?' column contains checkboxes, all of which are checked.

Item ID	Type	Name	Revision	Used by ID	Type	Name	Revision	Update?
145	PCB Footprint	C_mpc	2.0	146	Part	C_mpc2	1.1	<input checked="" type="checkbox"/>
			2.3	147	Part	C_mpc3	1.3	<input checked="" type="checkbox"/>
			2.1	148	Part	C_mpc4	1.3	<input checked="" type="checkbox"/>
			2.1	274	Part	C_mpc5	1.1	<input checked="" type="checkbox"/>
			2.0	275	Part	C_mpc6	1.0	<input checked="" type="checkbox"/>
			2.0	283	Part	C_mpc1	1.0	<input checked="" type="checkbox"/>
			2.0	355	Part	C_mpc7	1.0	<input checked="" type="checkbox"/>
			2.0	360	Part	C_mpc8	1.0	<input checked="" type="checkbox"/>

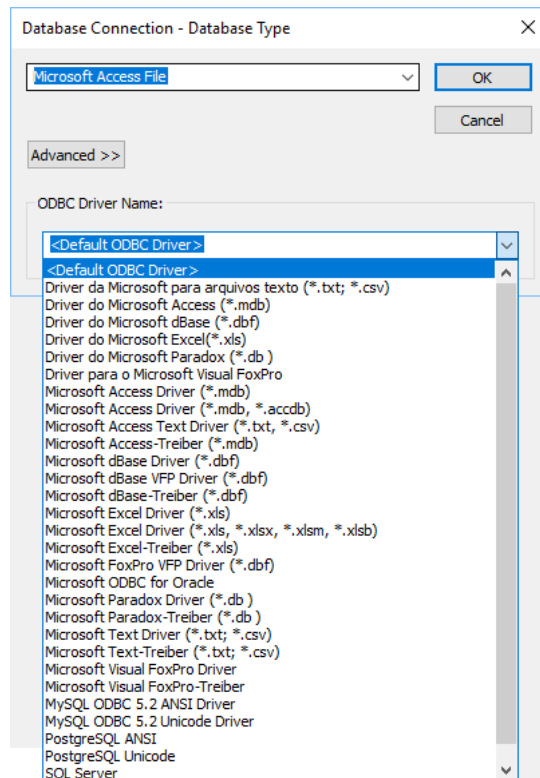
Database Connection Option – Database Type

The **Database Connection – Database Type** dialog now has an additional **Advanced>>** button that will display a list of all the ODBC Drivers installed on the system and allow an alternative to be selected. These drivers will be the default ones installed with the operating system and any that have been user installed afterwards.

Clicking the **Advanced>>** button reveals the ODBC driver list.



Choose the driver required from the drop down list.



Area Attribute Rules in Schematics

In a Schematic design, you can now add **Areas**, these areas can be assigned rules in the Technology. The rules can be assigned to Components, Pins or Nets in that area. As well as areas, the rules can also be assigned to a **Block** or **Block Instance**.

The attributes are passed to PCB on their respective Components, Pins or Nets using **Translate To PCB** and **Synchronise Designs** where you can then assign rules to those attributes.

This mechanism will enable you to quickly 'group' collections of items together and assign rules without the need to use attribute naming. This is particularly useful where for example, you might want the same rule assigned to Nets but which have different Net Names, like CLK, RESET, \$0901...etc. These can still be done but this is a much easier to allocate attributes and rules. Another advantage of using this process is that any additional items added to the area will automatically inherit the attribute name without needing to edit it.

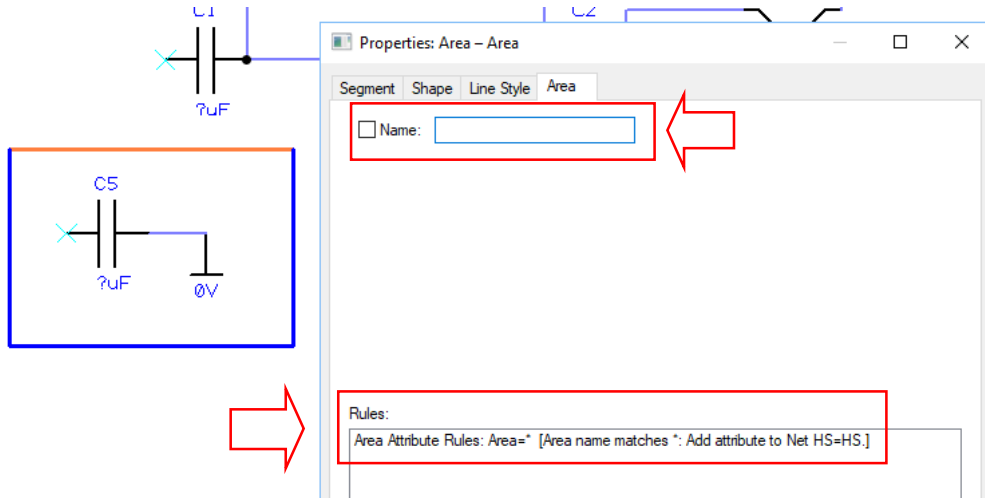
The Process Summary

The process for using this new feature will be:

- In **Schematics**, insert an Area (using **Insert Area**)
- In the **Technology**, use the new dialog under **Rules – DFM/DFT Area Attribute** to add a new rule.
- Under **Match Items**, choose **Area** as the **Item** (you can also choose, **Block** or **Block Instance** if you are not using an area) and set a name value to Match the name given to your Area. This can contain wildcard characters where * will match all names.
- Under **Add Attribute**, set **Add to** to the type of item (**Component**, **Net** or **Pin**) to which you want the attributes added, then add the name and value of the attribute to be assigned.
- Use **Translate to PCB** (or Synchronise Design). The translated (**Component**, **Net** or **Pin**) items will that match the rule will have the specified attribute name and value automatically added.
- In PCB, the attribute name is then available to be assigned to another rule(s).

Use in Schematic Editor

In the Schematic editor, use the new **Insert Area** option. You can optionally give it a name, this will help you identify and clarify rules. You would do this using the **Properties** dialog of the selected area.



The **Properties** dialog will display the **Rules** that have been assigned to the area.

The rules will be assigned from the **Technology** using the new dialog under Rules – DFM/DFT **Area Attribute**

Enable	Item	Match Value	Attribute		
			Add To	Attribute Name	Attribute Value
<input checked="" type="checkbox"/>	Area	*	Net	HS	HS

Match Items:

Item: Area

Match: *

Add Attribute:

Add to: Net

Attribute: HS

Value: HS

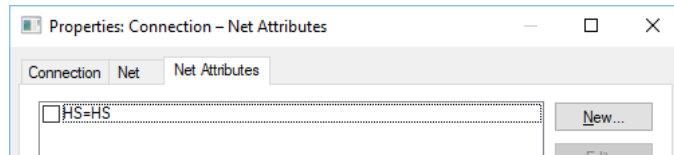
From the **Item**: drop down list you can choose **Area**, **Block** or **Block Instance**. An area can be named or * to choose all. As with all rules, you may wish to make a general ‘catchall’ rule for everything else [*] as the last item of the list to capture any unnamed rule. The match value can contain Wildcards. Choosing **Block** will enable hierarchical blocks to be selected. Choosing **Block Instances** means individual block names can be selected.

The Add Attribute **Add to**: is the rule that defines what type of items the attribute will be added to. Choose **Component**, **Pin** or **Net** from the drop down list. Any Component, Pin or Net which has a pin completely enclosed within the block or area will receive the attribute.

Finally, the rule defines which **Attribute** and **Value** will be added. The attribute will only be added if the item does not have that attribute assigned (either directly, inherited from a part, or from an area attribute rule define further up the list).

Use in PCB Editor

The area rules are assigned during **Translate To PCB** or the **Synchronise Design** options. Once in PCB you will create rules that match the attribute and name assigned to the area in the Schematic design. For example, a Net that has inherited an area attribute from the Schematic will show the **Attribute** and **Value** in its **Properties**.



Because the Net has this attribute, it can now be used for selection within a rule in the **Technology** dialog.

Plot Areas in Doc Symbols


With the introduction of **Areas** in the Schematic design, you can also add them to **Schematic Doc Symbols**. Once a symbol that uses these is placed in the design, this area can then be used as a plotting area in CAM Plots. For use in CAM Plots, the area must be **named** in the **Doc Symbol** editor. An area used in an SCM Doc Symbol cannot be used to define rules though, just CAM Plots.

Component Place Rules in PCB

A new page has been added under the **DFM/DFT Rules** in the **Technology** dialog for **Component Place**.

Enable	Attribute Name	Match Value	Side	Area	Keep In
<input checked="" type="checkbox"/>	<Component Name>	C*	All		<input checked="" type="checkbox"/>

Attribute:

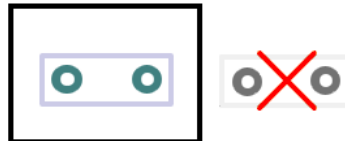
Match: 

On Layers

Side:

Within Areas:

Keep In:



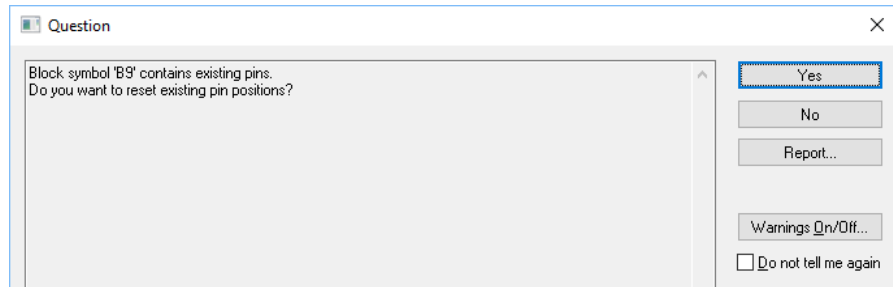
You can define rules which define which areas a Component must keep in/out. Components can be matched by attribute. In particular, you can use the Schematic area attribute rules to assign an attribute.

Hierarchical Block Instances

Regenerate Block Instance – Reset Pins

An option has been added to the existing feature, **Regenerate Block Instance**, to reset existing pin positions.

When this option is selected from the context menu, following the prompt of what action you'd like to take with the lower level **Unused Block Ports**, you are asked how to resolve the upper level **Block Pins**.

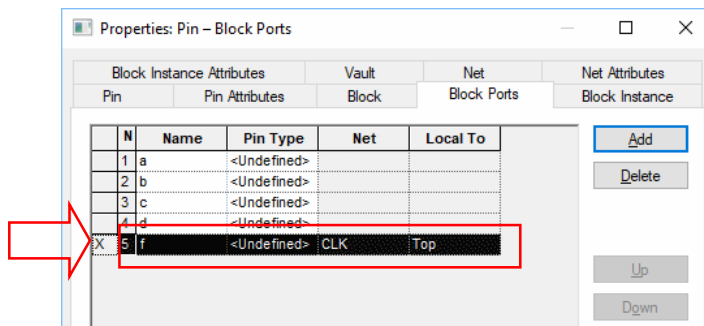


Selecting **Yes** will arrange all pins based on their type (e.g. input, output, etc.).

Selecting **No** will keep existing pin positions the same and arrange new pins to fill in the gaps (based on the Block Defaults), which will prevent existing pin connections from being disturbed.

Delete Block Port

You can now delete a **used Block Port** from the **Properties** dialog (provided the **Block Symbol** is not from a library).



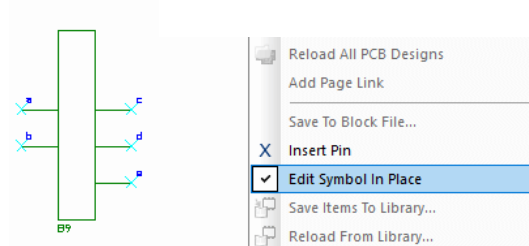
Block Instance – Edit Symbol in Place

Generated block symbols can now be edited within the schematic design in the same manner as they can be in a Schematic doc symbol window. This mode within the main Schematic editor means you don't need to go into another mode to make edits. It also means you have access to additional functionality such as Insert Pin, Move, etc.

18 Pulsonix Version 10.5 Update Notes

To use this edit mode, select the **Edit Symbol in Place** option from the context menu. You will be put into a special editing mode. Use the functionality available on the menus and the context menu. When finished press **Save** on the menu bar. You can also exit by deselecting the checked **Edit Symbol in Place** option from the context menu.

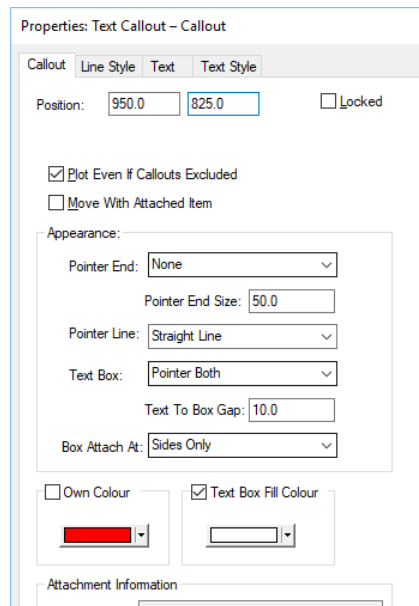
Note, this only works on **Generated Blocks** and not **Block Symbols**.



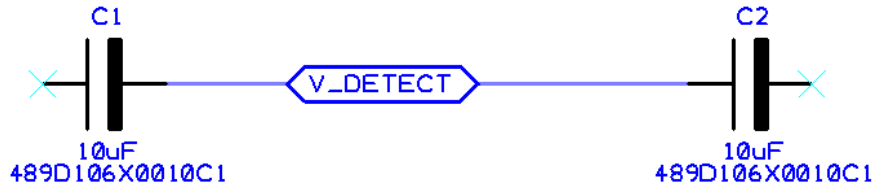
Text Callouts with Solid Colour Background

In Schematics, from within the **Properties** dialog of a **Text Callout** in the **Symbol Editor**, you can now define **Own Colour** and **Text Box Fill Colour** of a text callout.

With **Own Colour** left unchecked, the design background will be used to fill the callout.



This is useful if you wish to create a **Net Label** which can be placed on a **Net** to display the net name for that net and whether it is an Input, Output or Bi-Directional signal. Because dynamic net labels using a Signal Reference are created using a Doc Symbol and text callout, this feature can be used to blank out the background thus creating nicer looking schematic drawings.



A new **NET LABEL** Doc Symbol has been created and added to the SCM Doc Symbol Library for use when adding **Signal Reference** symbols.

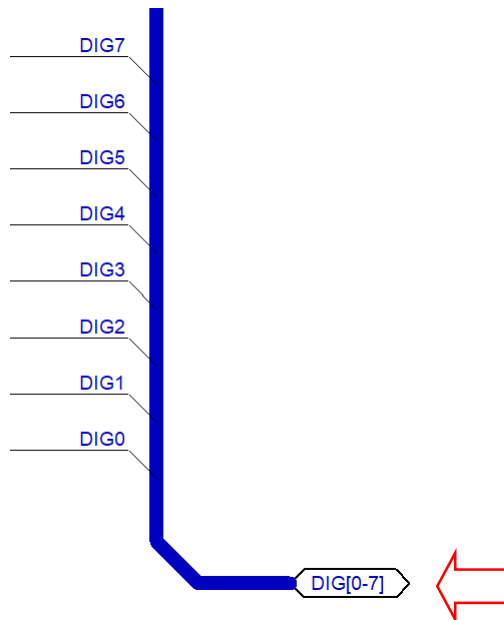
Text Callouts with Solid Colour Background In PCB

This also applies for an inserted **Layer Stack** in a PCB which now adopts the colour of the Text item that is attached to it. To do this, right click on the text callout from the **Layer Stack**, select **Properties**, **Callout** tab and then check the **Own Colour** and **Text Box Fill Colour** boxes.

New Doc Symbol Type - Bus Reference

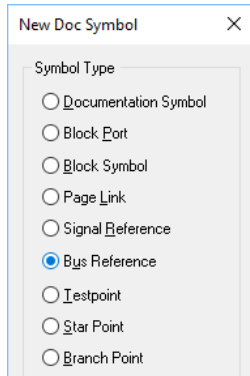
A new **Schematic Doc Symbol** type of **Bus Reference** has been added.

Bus References are similar to signal references and can be added to Schematic designs via the new menu command **Insert Bus Reference**. These are attached to buses and used to label the bus. They display the nets within the bus.



A bus can be started and ended on a Bus Reference. This can be configured within the symbol to display various details of a bus, for example, the Net Name.

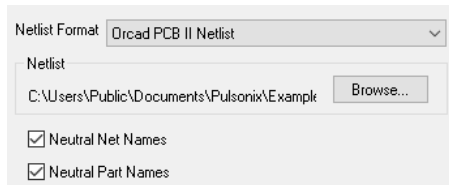
When creating a new Schematic Doc Symbol or editing an existing one, a symbol type of Bus Reference can be selected.



A Bus Reference can also be selected from the context menu when adding or editing a bus.

Orcad Netlist Export in SCM – Suppress Net/Part Names

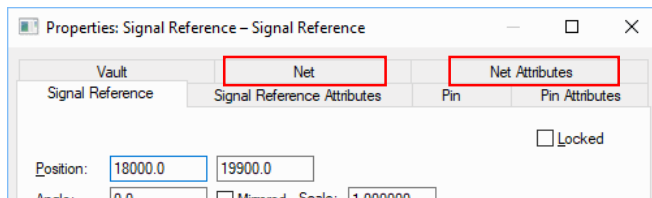
Two new switches have been added to the Export PCB Netlist option Schematics when the Netlist Format **Orcad PCB II Netlist** is selected. Use the **Neutral Net Names** and **Neutral Part Names** check boxes to neutralise **Net** and **Part Names** when exporting the design.



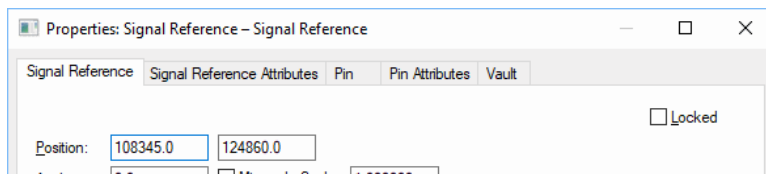
Selection of a Signal Reference with a Single Pin

A small change has been made so that when a **Signal Reference** with only a single pin is selected, more information is displayed in **Properties** about the pin and **Net** that is attached to it.

The dialog in V10.5 now looks like this:

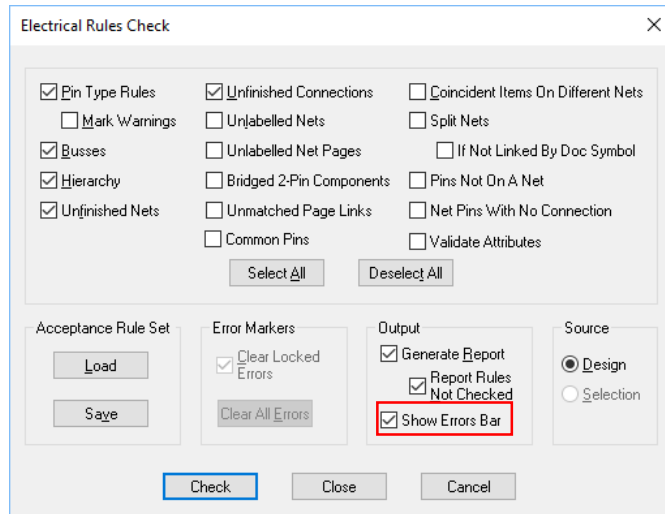


Previously in V10.0, the whole Sig Ref was selected with no net information.



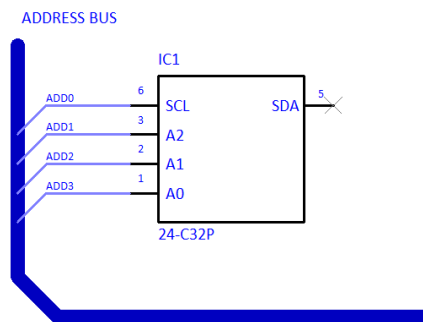
Electrical Rules Check - Show Errors Bar

The **Show Errors Bar** switch has been added to the **Electrical Rules Check** dialog. It will now show the ERC Errors bar when the **Show Errors Bar** option is checked and the check button is pressed.



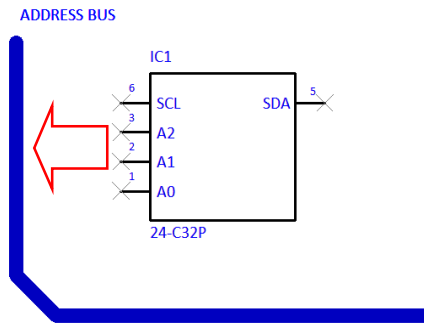
Auto Weld Component To Bus in Schematic Designs

You can now drag a Component in the Schematic design so that when the pins of the Symbol are over a bus segment it will weld it to the bus.

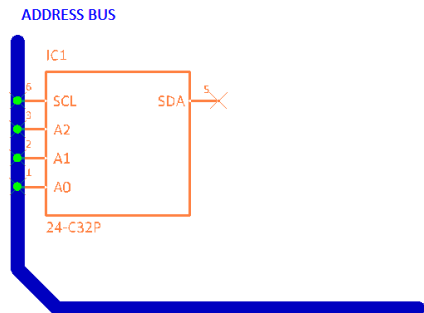


For Closed Buses

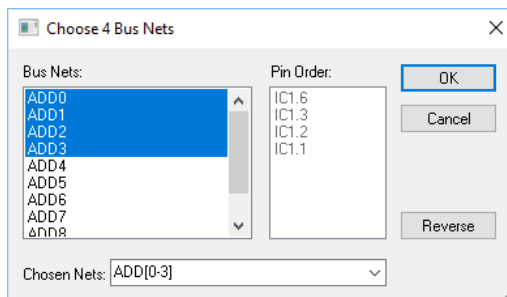
Move a Component or Gate (Symbol) so that the pins are coincident with the bus, ensure the pin grids and bus grid are the same:



Weld Markers between the bus and Component pins are displayed indicating the component can be welded:

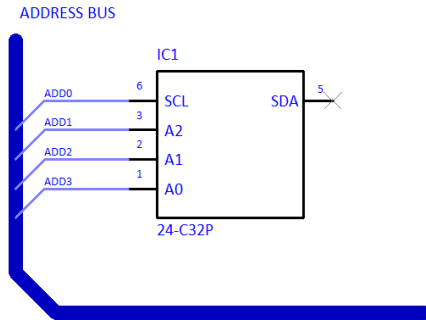


When a bus is closed (i.e. with named nets), a dialog will be presented to choose the net names from the bus for each of the pins:



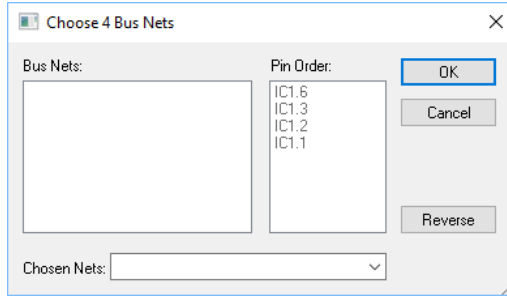
When choosing nets to add to the pins, it will always select the first matching number of nets/pins. However, the Bus Nets can be individually changed by picking them from the list, the Shift and Ctrl keys can also be used to aid selection.

Moving the symbol away from the bus will show the connections and bus terminals clearly.

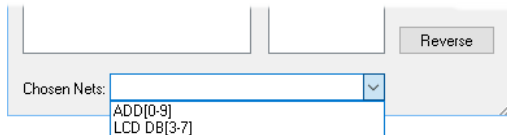


Open Bus

If the bus is an open bus you cannot pick from net names on bus, but can type the required net name range (or pick an existing closed bus name range from a drop down list).



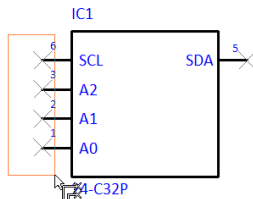
If other closed buses are defined within the design, you can choose the nets using the drop down list:



Multi Connect in Schematic Designs

A new option called **Multi Connect** enables you to start and 'route' multiple connections at the same time in Schematics.

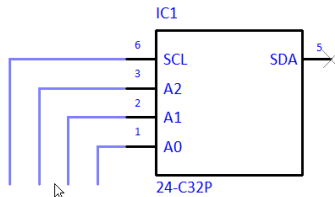
To enter this mode, select **Multi Connect** from the **Insert** menu or select it from the SCM toolbar.



24 Pulsonix Version 10.5 Update Notes

It has its own cursor where you can simply connect multiple symbol pins to another symbol or bus segment.

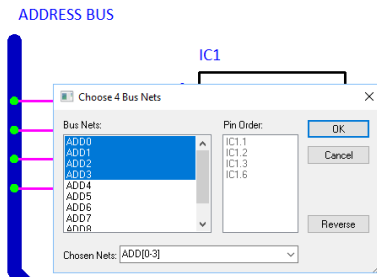
Once you start adding the multiple connections, clicking the mouse will add a corner.



When moving the cursor over a Component's pins or bus, weld spots will appear indicating that the connections can be finished. Click to finish the connections on that item.

Ending on a Bus

If you select to end on a bus segment and the connections are not already on nets in the bus you will be presented with a dialog to select the nets from the bus that you require.

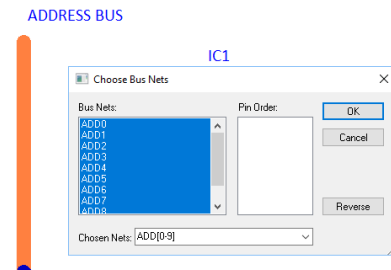


The pin names are displayed in the dialog to show the order for the net names. You can type a net range if required.

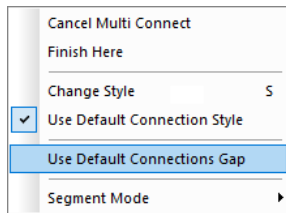
Starting from a Bus

Entering **Multi Connect** mode and selecting an open bus will allow you to type the net name stem you require. It will automatically select the correct nets and create a closed bus (if there were no other connections to the open bus).

In this mode you can pick a bus segment to start from and the dialog will be presented to choose which nets you wish to "pull" from the bus. Choose the **Bus Nets** required.

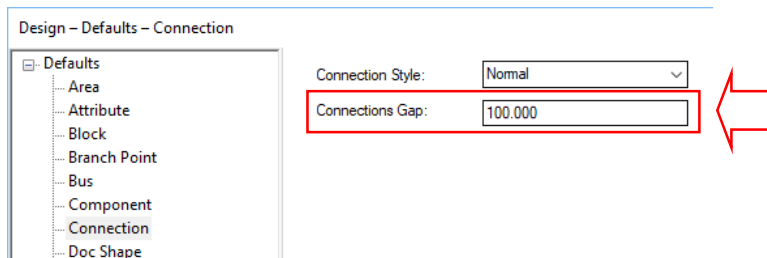


Once the connections have been started, you can right click and select **Use Default Connection Gap**. This is set in a new option on the **Connections** page of **Design Settings** page and enables you to bring the connections together.

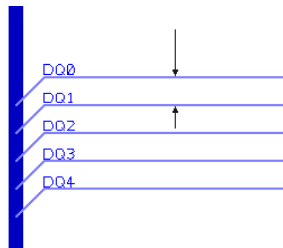


New Default Settings – Connections Gap

There is now a **Connections Gap** control in **Design Settings** under **Connections**.

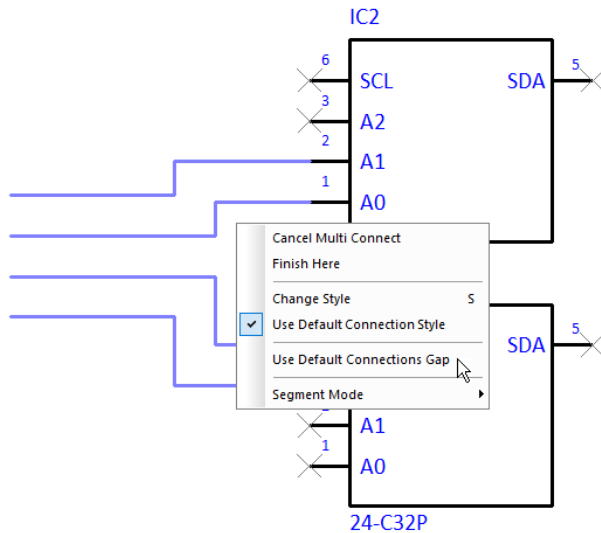


It is used by the new **Multi Connect** option for the gap between connections when starting from a Bus.



This value is also used for the distance between connections when gathering them together if the starting positions are unequal (as they would be if starting from different Component pins). Once the connections have been started, right click and select **Use Default Connection Gap** from the context menu.

The connections will then be gathered together.



Synchronise with Testpoints Changes

A change made to Test Points in V10.5 means that any pair of designs that fully synchronised in Version 10.0 and contain Test Points will now produce a differences report that says the connectivity is the same but best update the PCB for future matching performance. This is because Test Points in Schematics now have an ID and it needs to be copied to the PCB. The added ID is an internal identifier that means synchronise can more easily identify changes made to Test Points.

Design Settings switch for Synchronisation of Net Class Changes

There is a new switch for Synchronisation in Design Settings for a PCB design. The **Apply Net Class Changes** switch enables Net Classes that exist in the PCB to be ignored. For example, for imported designs, some import formats have Net Classes in the PCB but none in the Schematic, this switch enables you to ignore them.

PCB in Safe Mode

Allow PCB Only single pin nets

Synchronised Design Name

Name: Pcb1

Back Annotation

Enabled

Synchronise with Schematic

Apply All Rules Strictly

Apply Footprint Changes

Apply Net Class Changes

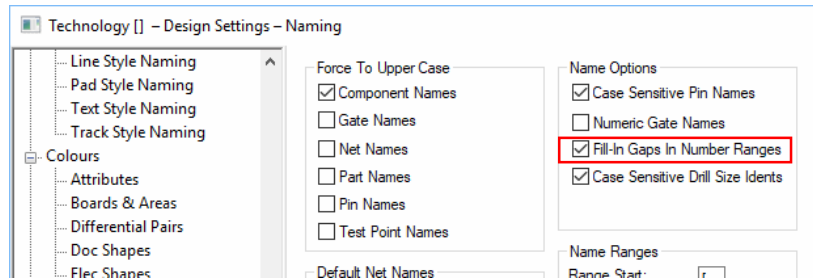
Ignore Attribute White Space

Allow Update of Schematic to match PCB

Naming – Fill-In Gaps In Number Ranges

There is a new option in the **Technology** dialog under **Design Settings, Naming** called **Fill-In Gaps In Number Ranges**. Enabling this option forces names for new **Components, Blocks, Block Instances, Test Points, Branch Points** and **Star Points** to fill in any gaps between name numbers before adding to the end of the number range.

Note, having the option enabled makes naming act as it did previously to V10.5. Disabling it will ensure names are always added to the end of the list.



Change Part – Default Selected Library

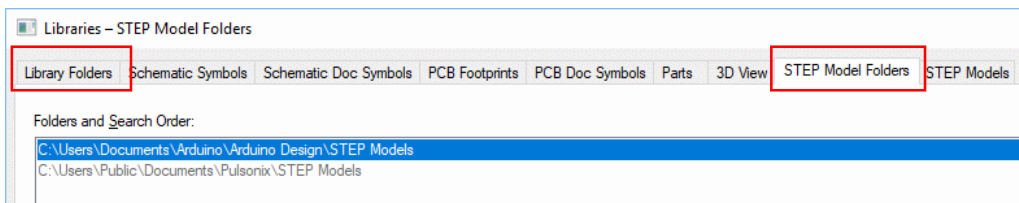
When using the **Change Part** dialog (for example, from **Replace Part** or **Component Properties**), the **Look In:** library now defaults to the library of the selected Part instead of **[All Libraries]**.

STEP 3D Preview Enhancements

STEP Model Folders

There is an additional tab in the **Library Manager** for **STEP Model Folders**. This makes the access to the STEP model file paths easier to access without leaving this dialog.

Also in the **Library Manager**, the original **Folders** tab has been renamed to **Library Folders**.



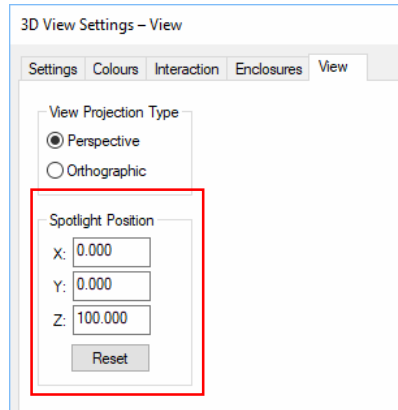
Spotlight Position in STEP 3D Preview

You can now adjust the position of the 'spotlight' in the **3D Settings** using the **View** page.

The **Spotlight Position** controls can be used for adjusting the directional spotlight used in the 3D STEP Preview to something other than its default X=0, Y=0, Z=100 position. For example, changing

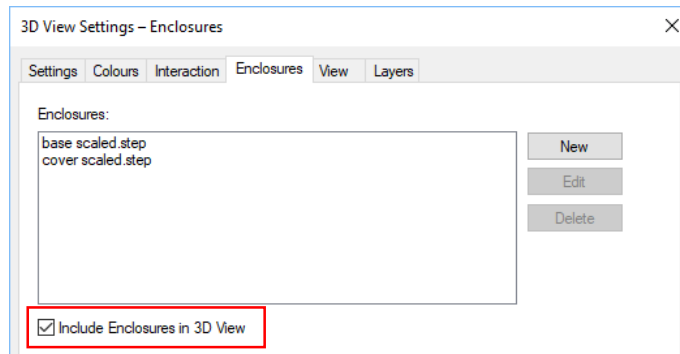
28 Pulsonix Version 10.5 Update Notes

it to 0, 0, -100 will shine it from below the board instead of above. The **Reset** button will also reset it back to the 0, 0, 100 position.



Include Enclosures in 3D View switch in STEP 3D Preview

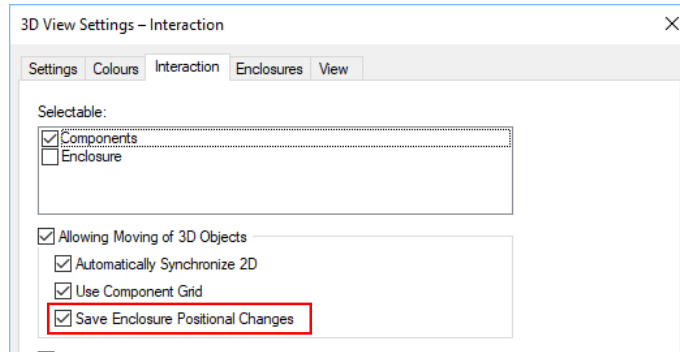
The new **Include Enclosures in 3D View** option, on the '3D View Settings, Enclosures page, allows you to choose whether or not to include the listed enclosures in the **STEP 3D Preview** or **STEP Output**.



New Save Enclosure Positional Changes option in STEP 3D Preview

The new **Save Enclosure Positional Change** option, on the **3D View Settings, Interaction** page, determines whether or not any manipulation of an enclosure model in the STEP 3D Preview is stored in the PCB design when the STEP 3D Preview window is closed.

Note, the 2D PCB design itself will need to be subsequently saved for the enclosure changes to be permanently saved. This option operates independently of the existing **Automatically Synchronise 2D** option.



Exploded View in STEP 3D Preview

You can now choose an **exploded** view representation of your PCB design in the **STEP 3D Preview**. This is a view of your board but with separation between the layers to show you the internal layer structure. This feature is enabled through the **3D Settings** dialog, **Settings** page and a check box, **Show Exploded View of Layer Stack**.

With this enabled, you can control the **Substrate Additional Thickness**, **Layer Drawing Thickness** and the **Gap For Exploded View**. For these settings, there is a **Units** selection from which you can choose one of the standard units to use.

Substrate Additional Thickness, this is the thickness of the substrate layers in the view which represent the board, the thickness specified here is added to any thickness defined in the Layer stack from the **Technology** dialog. If the layer thicknesses are specified in the Technology, you can set the value here to 0.

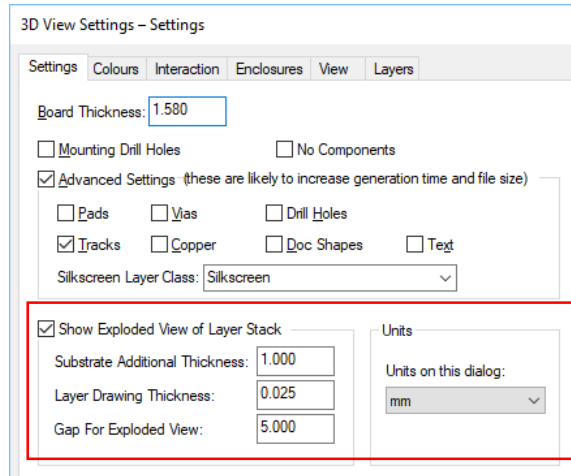
Layer Drawing Thickness, this is the thickness allowed for each layer of information (copper, silkscreen, etc.). This value should not be set too small (or to zero), otherwise each 3D layer will occupy virtually the same 'vertical' location and will 'bury' itself into the substrate. The end result would be that tracks and other items would disappear from the 3D picture, the default is set to 0.025mm for example.

Gap For Exploded View: this is the spacing to allow between substrates when drawing an **Exploded View** of a multi-layer board.

If an exploded view is selected in the Settings dialog, this view is what will also be written out to a STEP output file if one is generated.

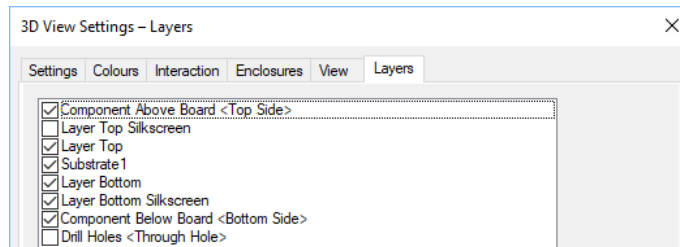
Enclosures are not shown in the exploded view. If you switch from a 3D Preview showing the enclosures to one that isn't, you will be prompted that the option will regenerate the view for you.

30 Pulsonix Version 10.5 Update Notes



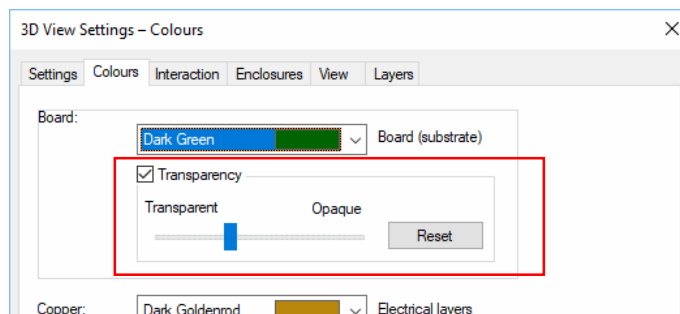
Layers Page in STEP 3D Preview

Also for use with the **Exploded View**, a **Layers** page is now available in the **Settings** dialog. Once the **STEP 3D Preview has been opened**, this allows specific layers to be switched off. If you change the Layer setting, you will be prompted that the option will regenerate the view for you.



Board Transparency option in STEP 3D Preview

The new **Board Transparency** option is available on the **3D Settings, Colours** dialog in both the exploded and non-exploded views. If you change the transparency setting, you will be prompted that the option will regenerate the view for you.

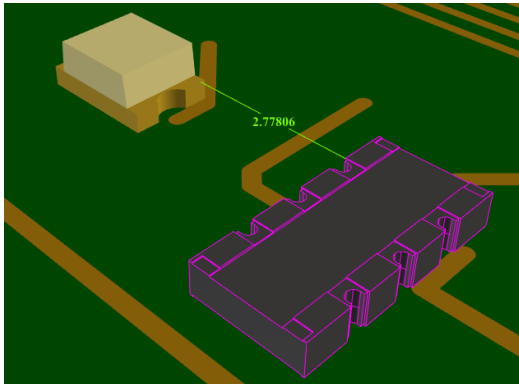


New Measure feature in STEP 3D Preview

There is now a new **Measure** option in the **Tools** menu which allows the distance between two 3D items to be shown on screen in the form of a measurement dimension.

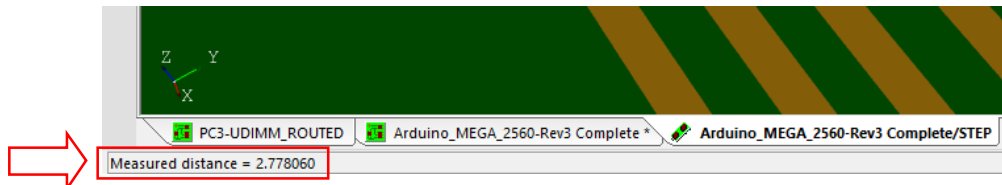
Note, to use this option, you must have **Components** and/or **Enclosures Selectable** in the 3D **Settings, Interaction** dialog.

Selecting the option will initiate measure mode. The measure cursor will be displayed and a message in the status bar will instruct you to select the item from which to begin measuring. By default, in measure mode, selection behaves as it does normally allowing complete 3D objects, such as components and enclosures to be selected for measuring between. It additionally allows the board substrate to be selected as a measurement item.



Holding down the <Shift> key while clicking changes the selection mode so that sub-parts of a 3D item, such as a particular face, edge or corner may be selected as a more exact point from which to make a measurement.

Once done, the selected part of item is highlighted white and the status bar will then instruct you to select the second item to which to measure to. Doing so will terminate measure mode and a measurement dimension will be displayed showing the distance in millimetres between the two selected items. Selection mode will return to selecting just whole objects and the cursor will return to the default arrow. The measured distance is also shown in the status bar.



If the two items chosen are touching, no measurement dimension will be displayed and the measured distance will be shown as zero in the status bar.

If one of the items measured between is subsequently moved, the dimension will update to show the new distance.

Selecting the **Measure** option again will remove the existing measurement dimension and restart measure mode.

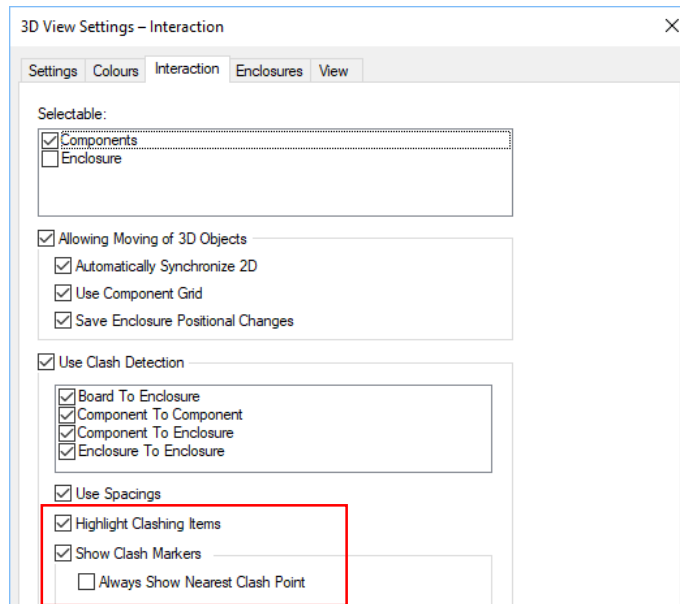
Pressing the **Esc** key while measure mode is active will exit without displaying a measure dimension.

The measurement dimension is not included the STEP output if the **Write STEP file** option is used.

Clash Markers in STEP 3D Preview

Clash Markers

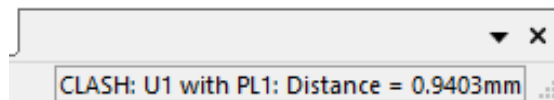
There are new switches on the **3D Settings** dialog under **Interaction** for use during clash detection in the **STEP Preview**; **Highlight Clashing Items** and **Show Clash Markers**



Highlight Clashing Items will enable you to choose whether clashing items are highlighted in red whereas previously this was the automatic result of using clash detection.

Selecting the **Show Clash Markers** check box instructs clash detection to insert **Clash** markers in the 3D Preview to indicate where clashes between items are occurring. Where a spacing has been violated, additional marker lines will point to the exact position on each item that is causing the clash.

Clicking on a **Clash Marker** in the 3D Preview will display the names of the clashing items and the distance between them in the right-hand end of the **Status Bar**.



With the **Show Clash Markers** check box enabled, you can now also select the **Always Show Nearest Clash Point** option. This directs clash detection to always show a Clash Marker at the closest possible point between two items. Leaving it un-checked means that when a Clash Marker is shown indicating a gap less than the specified spacing, it may not be the point at which the gap is smallest. As the items are re-positioned further apart the Clash Marker will gravitate towards the closest point until eventually it disappears. The trade-off for enabling “Always Show Nearest Clash Point” is that because it has to perform more comparisons it is slower in operation.

First and Next Clash Marker Commands

New **First Clash Marker** and **Next Clash Marker** commands have been added. This means it is possible to cycle through each of the current clash markers and zoom in on their location. These commands are available on the **Tools** menu and as a command that can be added to a shortcut key.

The **Detect Clashes** menu option which previous always performed clash detection on all eligible items will now act on just current item if one is selected. If not, it will act on all items as before.

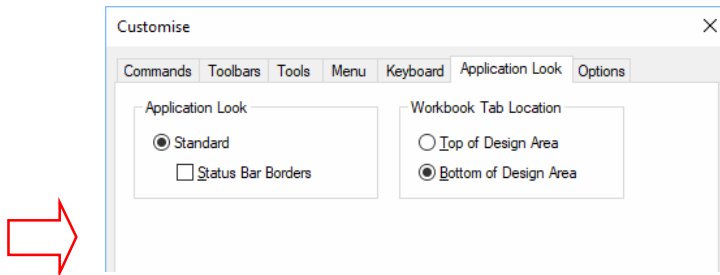
Support for 3D Space Mouse in STEP 3D Preview

If a **3D Space Mouse** is installed on your system it can be used to manipulate the viewpoint in the **STEP 3D Preview**. It provides an alternative to using mouse moves and button presses to move around the 3D View and unlike the mouse allows rotation in all three directions.

There are no settings in Pulsonix required. Once the mouse driver is installed it can be used with the Pulsonix product.

Changes to the Customise Dialog

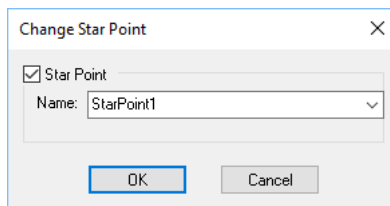
On the **Customise** dialog, **Application Look** page, the **Coloured** tabs selection check boxes have been removed as part of an ongoing interface update.



Star Points for other Pad Types in PCB

In PCB you can now make **Pads, Vias, Mounting Holes, Doc Symbols** and **Test Points** into **Star Points**.

To use this functionality, select the pad required and then choose **Change Star Point** from the context menu. You must select the **Star Point** check box and give it a unique name.



Once a pad on a net that already has a default Star Point is chosen, the pad will then become the Star Point and the existing default Star Point removed. You can make more than one a Star Point by selecting it and giving it a new Star Point name. If you use the same name of one already on that net,

the Star point status will be removed and applied to the new pad. If you remove the Star Point status altogether from a pad, a new Star Point using the default type will be added underneath the pad it was removed from.

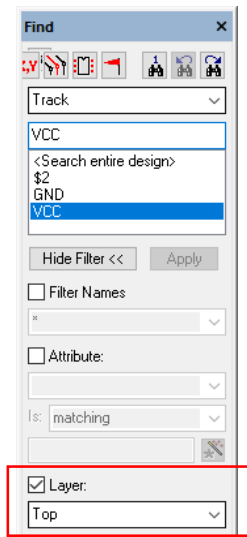
In safe mode, you will need an existing Star Point on the net attached to the pad (that Star Point would have come from the Schematic).

New Items in Find Dialog

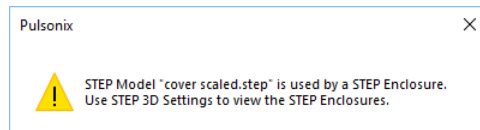
Three new Find categories have been added: **Find Track**, **Find Connection** and **Find STEP Model**.

Find Track (PCB) and **Find Connection** (SCM) finds the next Track or Schematic Connection on a chosen net.

Find Track can be filtered by **Layer**, so it can be used to select all tracks on a specific net and layer.

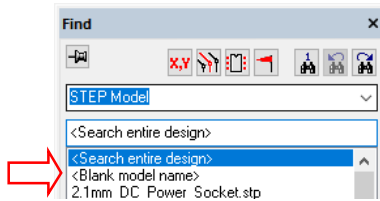


Find STEP Model can be used in a PCB design to find all Components, Doc Symbols or Enclosures that use a STEP Model. If an enclosure uses the chosen model, it cannot be selected. A message is presented saying a STEP Enclosure uses the model:



Existing STEP functionality allows you to suppress a STEP Model from the 3D Preview and output file by using a <STEP Filename> attribute on the Component without a STEP model being actually defined.

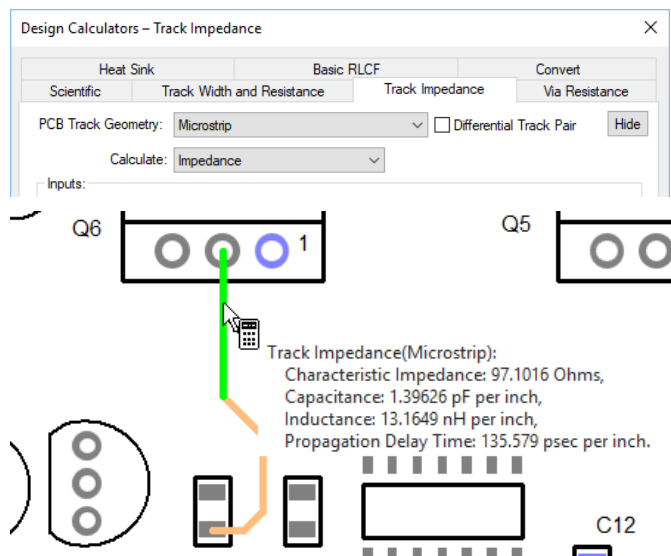
Within the **Find** browser when searching STEP Model names, the special model name <Blank model name> will be displayed and can be used to find Components or Doc Symbols that have the <STEP Filename> attribute, but with a blank value.



Track Impedance Tooltip Calculations

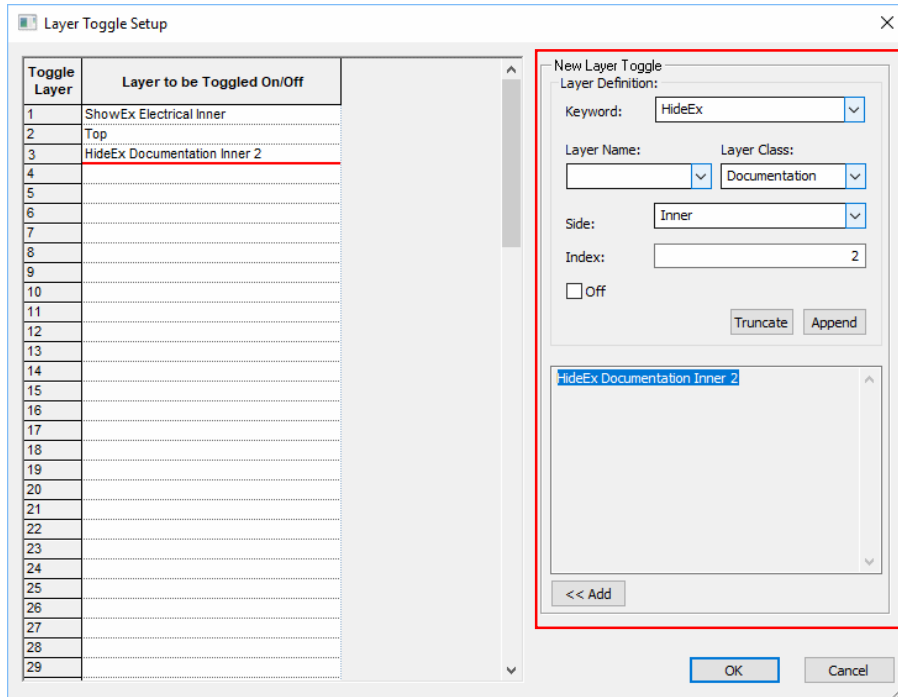
The **Design Calculator** utility and **Track Impedance** calculation can be used with a modeless dialog and the cursor used to 'probe' each track for inspection. Hovering over a track will show an active tooltip displaying the track impedance results. Other information regarding impedance is primed from the **Track Impedance** dialog.

To use this feature, run the **Design Calculator** utility and **Track Impedance** calculation. Press the Hide button and then move the cursor over the track you wish to inspect:



Set Toggle Layers Setup

New options have been added to the **Layer Toggle Setup** dialog for **New Layer Toggle** (available from the context menu on the **Layers Bar**). This adds a graphical interface to make layer choices as a simplified alternative to entering a toggle layer command by hand. You can select specific properties from a series of fields when adding/editing a toggle.



Using the Keywords Selection

If you want a Toggle Layer command to set a layer to be specifically visible or invisible rather than toggling between the two there are additional keywords that can be used as a prefix to the layer(s) specification. These **keywords** and their effect are described below:

Blank entry – this indicates that the layer will be toggled.

Show - the specified layer(s) will always be made visible.

Hide - the specified layer(s) will always be made invisible.

ShowEx - the specified layer(s) will always be made visible and all other layers will be made invisible.

HideEx - the specified layer(s) will always be made invisible and all other layers will be made visible.

Toggle - is provided for completeness and has the same effect as no prefix keyword. The specified layer(s) will toggle between being visible and invisible.

For example, defining Toggle Layer 3 as "ShowEx Electrical Top, Electrical Bottom" will allow the top and bottom electrical layers to be shown exclusively with all other layers hidden.

Layers, Layer Class and Side

The set of controls provided on the right side of the dialog allow a new Layer Toggle command to be constructed or an existing Layer Toggle command to be edited.

Either a **Layer Name** or **Layer Class** must be selected to create a valid toggle layer command and specifies what layer(s) the command is to act on.

As layer names can vary between designs it may be preferable to associate layers by characteristics rather than their explicit names.

Side provides the additional optional field which allow for more detailed control when a layer class toggle is being created.

Omitting **Side** and just specifying a Layer Class implies ALL layers of that class regardless of where they appear in the layer stack.

Index is available if the **Side** is selected as **Inner**. This will be a numeric value that refers to the Inner layer in the sequence it appears (in the Inner Layers list). It might be a more suitable alternative choice when specifying a specific inner layer (rather than all inner layers) to use a named layer command instead.

Omitting the **Index** when specifying **Inner** implies ALL inner layers, so "Electrical Inner" will toggle all the inner electrical layers.

The Layer Name, Layer Class and Side are defined in the **Technology dialog**.

Switching Multiple Layers On And Off Simultaneously

Use the check box **Off** to visibly switch a layer the opposite way to the first layer specified.

Where multiple layers are specified for a single **Toggle Layer** command, it is possible to simultaneously turn some of them on and some of them off thus allowing 'sets' of layers to be visibly switched. To achieve this, any layer to be switched the opposite way to the first layer specified should be followed by the word **off**. For example, defining Toggle Layer 3 as "Electrical Top, Electrical Bottom, Electrical Inner off" will allow the top and bottom electrical layers to be made visible and at the same time, all inner electrical layers to be made invisible. Using the Toggle Layer 3 command again will have the reverse effect.

Dialog Buttons

Selecting an empty **Layer Toggle** slot in the list, allows a new command to be entered. The current slot will be underlined in red, indicating it is ready to have a new command entered (or and existing command edited).

Once a command has been defined in the **Toggle Layer** selection, the <<**Add** button will insert the new command in to the currently selected empty slot.

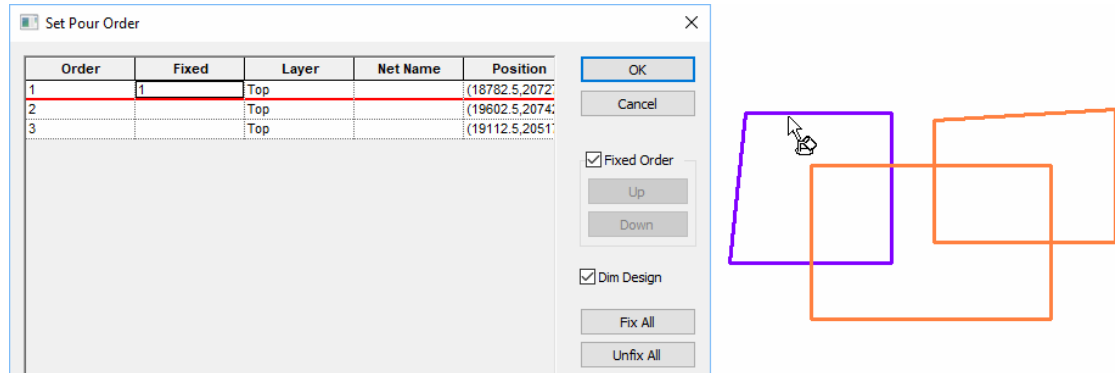
If a current **Layer Toggle** command on the left side is selected, you can either edit the command directly, or use the edit panel to the right side. The **Layer Toggle** controls will be primed with its details ready for modification. Once edited, the <<**Update** button will put a modified command back into its appropriate slot.

The **Append** and **Truncate** buttons allow Layer Toggle commands with multiple layer definitions to be created; **Append** will complete the current layer definition and start the next one and **Truncate** will delete the current layer definition and switch to the previous one or to an empty definition if there is not one.

Set Pour Order

Set Pour Order now a Modeless Dialog

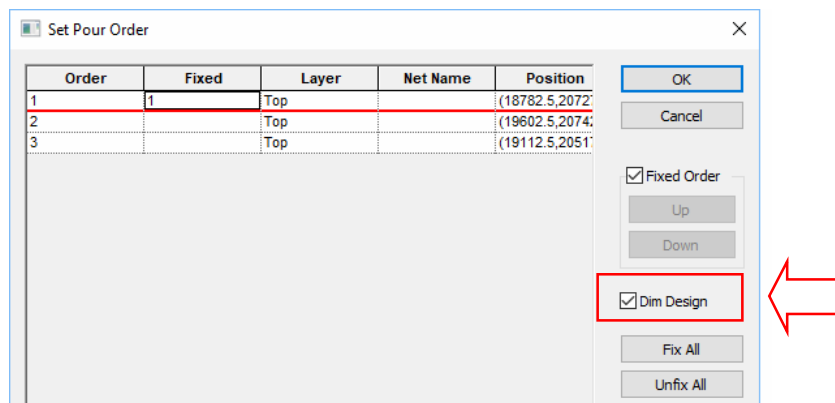
The **Set Pour Order** dialog has been updated to a modeless dialog so the design can be viewed (Pan, Zoom, etc.) whilst the dialog is open. This is available from the **Utilities** menu, **Set Pour Order** option.



Also, another change is Templates listed in the dialog can be selected in the design which highlights them in the dialog – this is a more interactive way for you to select templates.

Design can now be Dimmable during Set Pour Order

A check box has been added in the **Set Pour Order** dialog to **Dim** or not **Dim** the design when using this feature. If using multiple or complex pour areas, you can dim all other items to aid visualisation of the currently selected pour area.

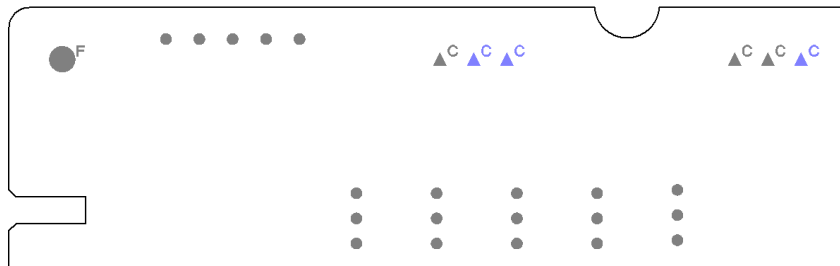


Drill Drawing Layer & Idents

Drill Drawing Layer

You can now display a drill drawing layer which will display the drill symbol and drill id in the design. These layers can be plotted instead of the existing drill drawing output.

Diam (thou)	Qty	Id	Plated	Symbol
30.0	1	E	Y	*
35.0	49		Y	●
42.0	9	C	Y	▲
78.7	4	F	Y	●



You can also add other items such as the drill ident table to this layer as well.

The Drill Drawing layer is defined as a new type of layer class, then used with a new Drill Drawing layer.

There is a new **Layer Type** to use called **Drill Drawing**. This is used to create the layer class used for a new Drill Drawing layer in the **Layers** page.

Name:

Used:

Layer Type:

Pad Oversize: <input type="text" value="0.000"/> <input checked="" type="radio"/> Absolute Size <input type="radio"/> Percent of Pad Size Min Undersized Pad: <input type="text" value="0.000"/>	Pad Types: <input checked="" type="checkbox"/> Component Pads <input checked="" type="checkbox"/> Doc Symbol Pads <input type="checkbox"/> Bgnd Pads <input checked="" type="checkbox"/> Free Pads <input checked="" type="checkbox"/> Mounting Holes <input checked="" type="checkbox"/> Comp Mount Holes <input checked="" type="checkbox"/> Vias <input checked="" type="checkbox"/> Micro-vias	Pad Condition: <input type="checkbox"/> Surface Mount Through Hole: <input checked="" type="checkbox"/> Plated <input checked="" type="checkbox"/> Non Plated <input type="checkbox"/> Only If Testpoint <input type="checkbox"/> Exception Only <input checked="" type="checkbox"/> Board Visible <input type="checkbox"/> Merge Cutouts	Areas Visible Component <input type="checkbox"/> Body <input type="checkbox"/> Clearance <input type="checkbox"/> Others Board Cutouts <input type="checkbox"/> Unplated <input type="checkbox"/> Plated	Break Shapes <input type="checkbox"/> Break Around Pad <input type="checkbox"/> Break Around Text <input type="checkbox"/> Break Comp Shapes At Board Gap: <input type="text" value="0.000"/> Draw/Plot Appearance <input checked="" type="checkbox"/> Pad Land <input type="checkbox"/> Drill Hole	Variant Components <input type="checkbox"/> Items Visible In All Variants <input type="checkbox"/> Use Variant Style Panel <input type="checkbox"/> Check Items Outside Board <input checked="" type="checkbox"/> Draw Shapes in Panel Design
---	---	---	--	--	--

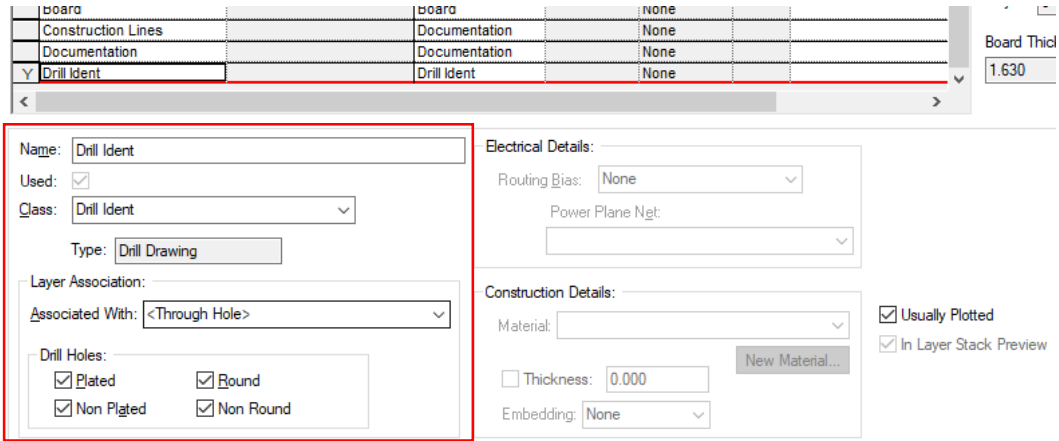
When using the Drill Drawing Layer Class, many items are preselected for use. You can also add other useful items to document your drill drawing, such as the **Board Outline** or **Areas**.

The Layer itself is relatively straight forward. You should define a **Name** for the layer and use the new **Drill Drawing layer class**. The **Type** will inform you that it is a Drill Drawing.

40 Pulsonix Version 10.5 Update Notes

You can create more than one Drill Drawing layer, for example, if you wish to have a drill drawing per layer span for your design showing layer spans other than <Through Hole>. To do this, create a new layer with a new name, use the Drill Drawing layer class and then choose the layer span that it will be **Associated With** using the drop down list of layer spans.

The **Drill Holes** check boxes allow you to select the type of drill hole you wish to view on the drill drawing.



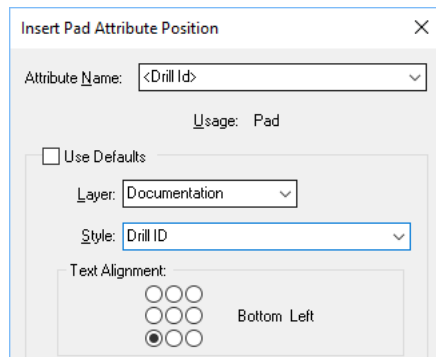
You will only see the full effect of this layer if you switch the drill drawing layer on by itself. If you don't, drill letters can be drawn underneath other items such as copper pads and will be obscured.

Display of Drill ID using Attribute Position

You can now display a drill id using the attribute position <Drill Id>. This is different from the new Drill Drawing Layer functionality and is not related.

This is an additional mechanism that allows you add a Drill Id Attribute Position to a pad. The attribute position can be moved individually and added to a different layer. You can also change the Style of this attribute. This may be required when producing a special plot of a layer combined with another layer.

Note, the <Drill Id> attribute will not be displayed if the **Drill ID** in the **Drill Size Id** table in **CAM Plots** or **Technology** is set to be not displayed.



Coloured Drill Sizes & Drill Table

You can now put a **Colour** on a **Drill Size**. When used, the drill ID and symbol are drawn in that colour. Add the colour using the **CAM Plots** dialog and **Drill Sizes** or define it in the **Technology** file. Select the **Colour** from the drop down list. Selecting **Use Own Colour** will force that colour to be used. If not selected, it will use the default colour.

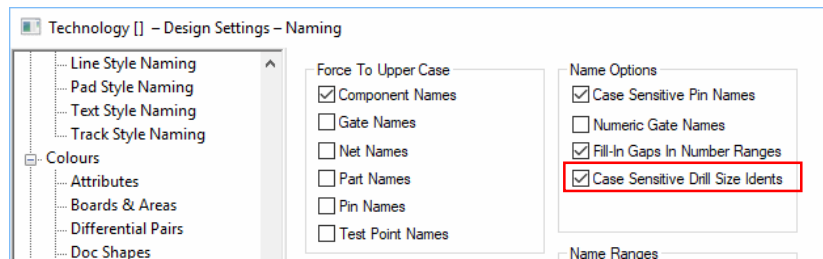
Used	Diameter	Plated	Id	Symbol	Symbol Size	Symbol Filled	Count	Min Diam	Max Diam	Description	Use Own Colour	Colour
Y	9.8	Y	A	Round	9.8	<input checked="" type="checkbox"/>	28				<input checked="" type="checkbox"/>	Red
	20.1	Y	B	Round	20.1	<input checked="" type="checkbox"/>	0				<input checked="" type="checkbox"/>	Grey
	35.0	Y	C	Round	35.0	<input checked="" type="checkbox"/>	0				<input checked="" type="checkbox"/>	Blue
	98.4	Y	D	Round	98.4	<input checked="" type="checkbox"/>	0				<input checked="" type="checkbox"/>	Green

The drill ID and symbol in the **drill table** are drawn in that colour.

Diam (thou)	Qty	Id	Plated	Symbol
16.0	151	A	Y	•
33.5	52	B	Y	•
37.4	52	C	Y	•
39.4	4	D	Y	•
93.7	2	I	Y	•
126.0	4	J	Y	•
Special	3	K	Y	•

Case Sensitive Drill Size Idents

From within the **Technology** dialog and **Design Settings, Naming** page, you can now force **Drill Size Idents to be case sensitive** by selecting the option.



Panel Editor - Tab Rout Changes

Additional **Design Level Spacings** have been added for **Panel** designs. You can now set minimum spacings for **Tab-Rout to Panel Edge**, **Tab-Rout to V-Score** and **V-Score to Drill Hole**. These will be used when checking Tab-Routs and V-Scores with **Design Rule Check**.

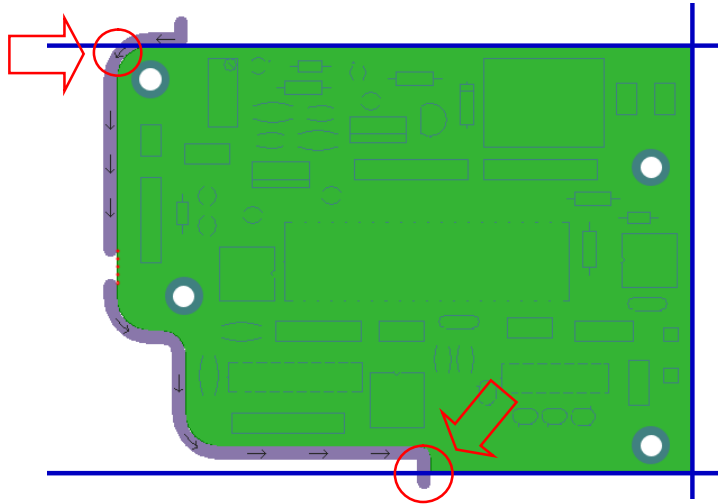
In a Panel design, the existing **Create Tab-Rout Around PCB** function will now obey these new Panel Design Rules and back-off or split the Tab Rout to avoid the panel outline, V-Scores, Tab-

42 Pulsonix Version 10.5 Update Notes

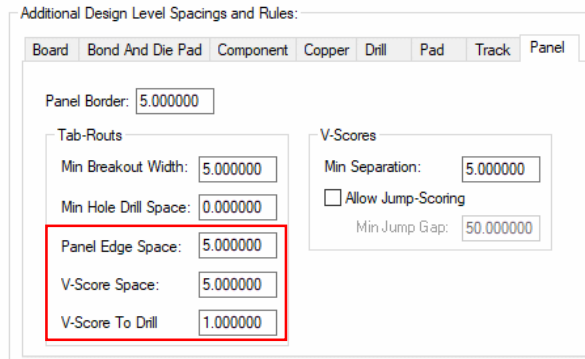
Routs and Other PCB instances. If possible, drill holes (mouse bites) will be optionally added within the backed off gaps (tabs).

When using **Create Breakout Tabs** to insert a tab with drill holes (mouse bites) in a curved Tab-Rout segment, the holes will now follow the contour of the curve in the board outline.

This new functionality will avoid this situation:



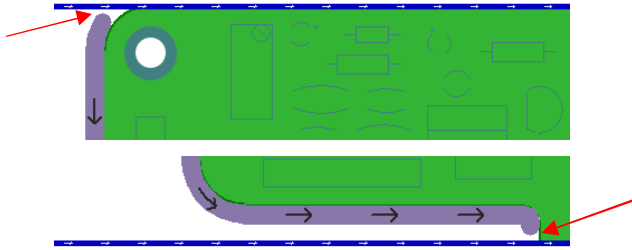
Additional settings have been added to the **Panel Technology** dialog under **Design Rules** and the **Panel** tab. New rules have been added for **Panel Edge Space**, **V-Score Space** and **V-Score To Drill** (space).



Panel Edge Space – this defines the minimum space required between Tab Routs and the edge of the panel. This will enable you to detect whether the Tab Rout is routing outside the panel outline.

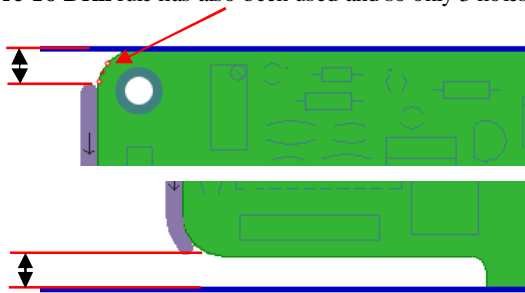
V-Score Space – use this rule to define the minimum space required between Tab Routs and V-Score lines.

The effect of this is illustrated below. The image has been cut to show the two areas of interest. Using a 1mm **V-Score Space** you can see the back-off gap.



If the **V-Score Space** spacing is set to a larger value, the effect will look more like the example below.

You'll also notice in the example below that because the Tab Rout has been backed off from the V-Score line, mouse bite holes have been inserted. Although the rule defined specifies 5 holes, the **V-Score To Drill** rule has also been used and so only 3 holes have been inserted.



V-Score To Drill – this rule defines the minimum V-Score line to drill spacing. This is particularly useful if you have a Tab Routs that use 'mouse' bites that are placed near to a V-Score line. It will also be used as your spacing rule when you insert a Tab Rout on a board outline. If mouse bit holes are too close, they will be removed (as shown in the example above).

Changes to Technology Files

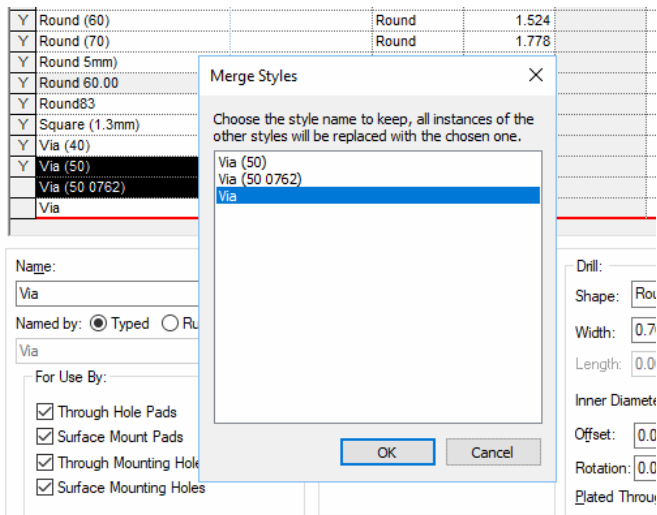
Merge Styles in Technology Dialog

From within the **Technology** dialog, on any other **Styles** pages, you can merge styles.

To activate this dialog, select more than one style and press the **Merge** button. This works both used and unused styles e.g. if you merge a round via with a square pad, if the round via is used in the design, it will be replaced by the square pad.

Y	Round 5mm	Round	5.000	2.000
Y	Round 60.00	Round	1.524	0.813
Y	Round83	Round	0.559	0.000
Y	Square (1.3mm)	Square	1.300	0.000
Y	Via (40)	Round	1.016	0.610
Y	Via (50)	Round	1.270	0.762
	Via (50 0762)	Round	1.270	0.762
	Via	Round	1.270	0.762

Pressing the **Merge** button reveals this dialog:



Net Names Page – Delete Multiple Rows

The ability to delete multiple Net Names has been added to the **Net Names** dialog in **Technology**.

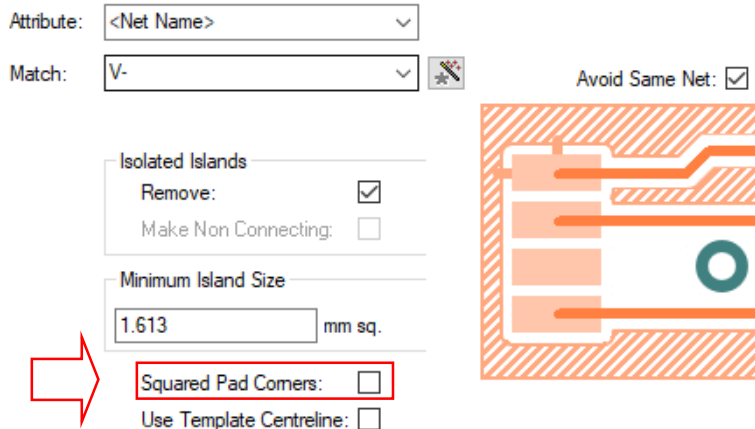
For the 'Delete' option button to become enabled, only Unused styles must be selected.

As well as 'dragging' down the list in the 'used' column on the right hand side, you can also multi-select using the Shift and Ctrl keys.

Name	Net Class	Type	Guard Space	Check Same Net	Use Own Colour	Colour	Display Connection
LCL_EN		Signal	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>
A2		Signal	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>
N070		Signal	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>
D0		Signal	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>
D2		Signal	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>
D4		Signal	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>
VCCIO1		Power	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>
VCCIO2		Power	0.000	<input type="checkbox"/>	<input type="checkbox"/>		<input type="checkbox"/>

Pour Squares Off Pad Corners

When pouring, you can optionally square off around pad corners. The **Squared Pad Corners** option is available in the **Technology** under **DFM/DFT - Copper Pour**.



When checked, this will cause the poured copper to have sharp squared corners **around rectangular pads**. If not checked, the copper will be rounded around the corners to maintain the precise spacing.

The images below show the effect of not checking and checking this option:



The difference is subtle but can be seen on all three of these rectangle pad examples below with the option selected:



Square-Ended Thermal Spokes

When using the **Copper Pour** option, if necessary, pour will **insert square ended spokes** to fit where a round ended spoke would not fit. This is a default option and works as a last resort when a rounded spoke style does not work, this setting is hard wired and cannot be changed.

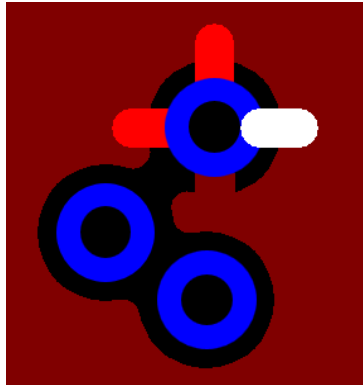
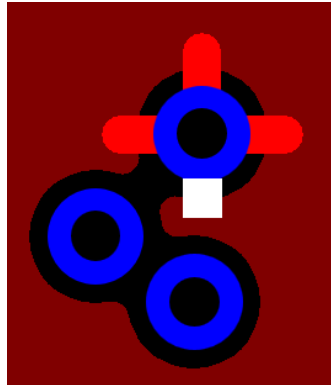


Image highlighting 'normal' rounded spoke



Same image but square spoke highlighted where the rounded spoke cannot be fitted

Use Centreline of Template for Pouring

When pouring, you can optionally use the **centre line** of the **Template** as the extents of the copper. The **Use Template Centreline** option is available in the **Technology** under **DFM/DFT - Copper Pour**.

Attribute:

Match:

Avoid Same Net:

Isolated Islands

Remove:

Make Non Connecting:

Minimum Island Size

mm sq.

Squared Pad Corners:

Use Template Centreline:

A cross-sectional diagram of a PCB showing a copper pour (orange) on a template (orange with diagonal lines). The copper pour is shown filling a gap between pads, with a central hole. The diagram illustrates how the copper pour's extent is defined by the template's centerline.

When checked, will cause the edge of the poured copper to run along the centre line of the template, this will mean that the centre line will always define the extents of the copper regardless of the thickness of the style used. If not checked, the centre line of the poured copper will follow the centre line of the template, so the extents of the copper will go to the full thickness of the template.

Ability to Enable/Disable Rules

The following affects all types of rules including **Net Styles** and **Match Pair Spacing** rules:

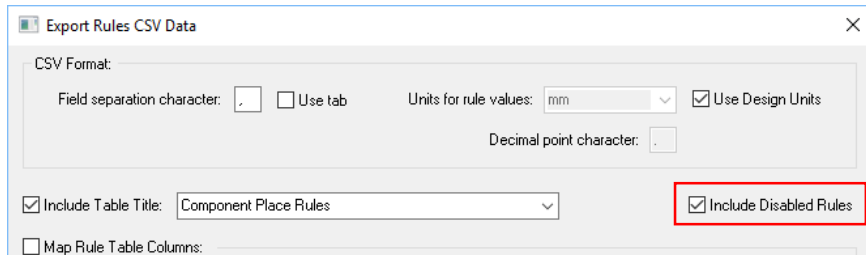
An **Enable** column and check box has been added so that rules can be switched off if not required. A disabled rule will not be used.

Enable	Attribute Name	Match Value	Avoid Same Net	Min Island Size	Squared Pad Corners
<input checked="" type="checkbox"/>	<Net Name>	*	<input type="checkbox"/>	1.613-	<input type="checkbox"/>
<input checked="" type="checkbox"/>	<Net Class Name>	+5V Override1	<input type="checkbox"/>	0.000	<input type="checkbox"/>
<input checked="" type="checkbox"/>	<Net Class Name>	+5V Override2	<input type="checkbox"/>	0.000	<input type="checkbox"/>
<input checked="" type="checkbox"/>	<Net Class Name>	GND Override	<input type="checkbox"/>	0.000	<input type="checkbox"/>
<input checked="" type="checkbox"/>	<Net Class Name>	GND Override	<input type="checkbox"/>	0.000	<input type="checkbox"/>
<input checked="" type="checkbox"/>	<Net Class Name>	NS1 Override1	<input type="checkbox"/>	0.000	<input type="checkbox"/>

New **Report Maker** commands have been added to report this status, see the *Report Maker Changes* section later on.

Selection of Disabled Rules in CSV Export

Within the **Export CSV** of rules dialog on the **Technology** dialogs, you can now opt to export rules that are **Disabled** using a new check box.



When exported, the CSV contains new column to show whether the **Enable** status is on or off (shown as 1 or 0).

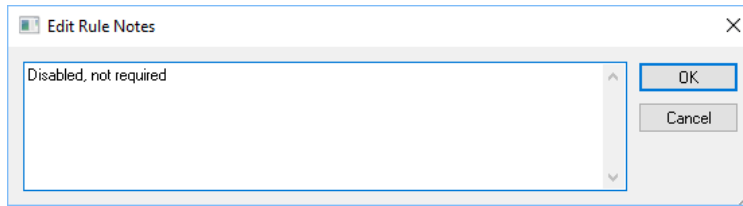
Ability to add Notes to a Rule

The ability to add **notes** to a rule has been added in Pulsonix V10.5.


These can be added by right clicking selecting **Notes** from the context menu.

Enable	Attribute Name	Match Value	Avoid Same Net	Min Island Size
<input checked="" type="checkbox"/>	<Net Name>	*	<input type="checkbox"/>	1.613-
<input checked="" type="checkbox"/>	<Net Class Name>	+5V Override1	<input type="checkbox"/>	0.000
<input checked="" type="checkbox"/>	<Net Class Name>	+5V Override2	<input type="checkbox"/>	0.000
<input checked="" type="checkbox"/>	<Net Class Name>	GND Override	<input type="checkbox"/>	0.000
<input checked="" type="checkbox"/>	<Net Class Name>	GND Override	<input type="checkbox"/>	0.000


Editing the Notes gives you a dialog into which you can type.



If a rule has notes, a small green triangle appears at the top right corner of its **Enable** cell.

Enable	Attribute Name	Match Value	Avoid N
<input checked="" type="checkbox"/> 	<Net Name>	*	
<input checked="" type="checkbox"/>	<Net Class Name>	+5V Override1	
<input checked="" type="checkbox"/>	<Net Class Name>	+5V Override2	

Hovering over the Enable cell will show the notes in a **tooltip**.


Enable	Attribute Name	Match
<input checked="" type="checkbox"/> 	<Net Name>	*
<input checked="" type="checkbox"/>	<Net Class Name>	+5V C
<input checked="" type="checkbox"/>	<Net Class Name>	+5V C
<input checked="" type="checkbox"/>	<Net Class Name>	ND I

Notes:
Disabled, not required


The **Report Maker** command **Notes** can now be used when reporting rules, see the *Report Maker Changes* section later on.

Cell Notification of Rule Errors

If a rule has a potential error, a **red warning triangle** appears at the top left corner of its **Enable** cell.

Enable	Item 1		
	Attribute Name	Match Value	Attribu
<input checked="" type="checkbox"/> 	<Net Class Name>	*	<Net Class Na

Hovering over the **Enable** cell will show the warning in a tooltip (after its Notes if it has any). For example if it uses a Net Class that no longer exists.

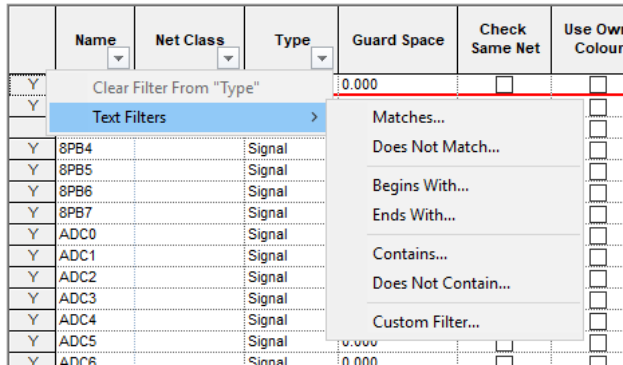
Enable	Attribute Name	Match Value
<input checked="" type="checkbox"/> 	<Net Class Name>	*

Warnings:
Warning: No spacing check areas currently match.

Grid Column Filters on Net Names Page

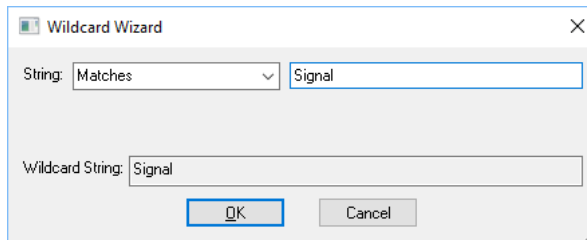
Excel-style wildcard-based filters have been added to the **Net Name**, **Net Class** and **Type** columns in the **Net Names** page grid in **Technology**. This allows the nets being shown to be refined.

The filters operate with wildcards, which are created using the **Wildcard Wizard**.

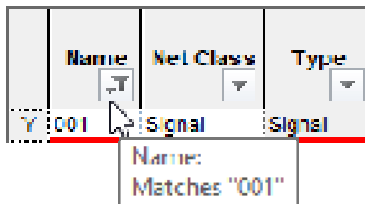


Clicking on one of the ‘down’ arrows on the column header will display a context menu. If a text filter has been defined, the **Clear Filter From ‘xxx’** will be available. If creating a new filter, it will be greyed out. In the example above, the filter is for ‘Type’.

Selecting one of the filter words in the list will display the **Wildcard Wizard**.

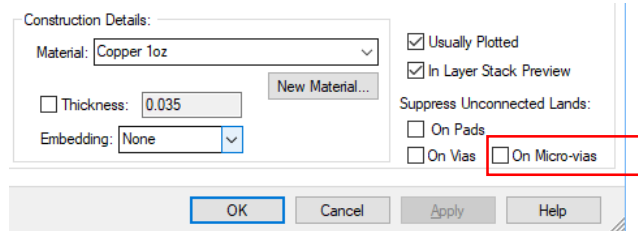


When a filter is set, the filter button changes to reflect this. Hovering the cursor over the filter button will show a tooltip and the filter applied.



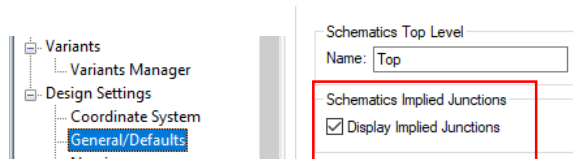
Suppress Unconnected Lands for Micro-Vias

Within the **Technology** dialog and **Layers**, you can now **Suppress Unconnected Lands** on **Micro-vias** separately to normal Vias.



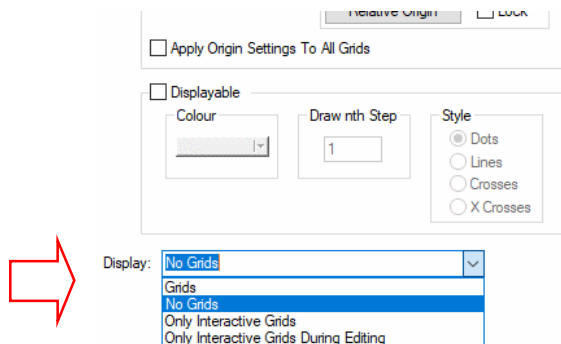
Display Implied Junctions moved into Technology

The **Display Implied Junctions** option has been moved from the **Options** dialog to the **Schematic Technology, Design Settings** and **General/Defaults** page. This means that each design now has this flag rather than it being a system option.



Changes to the Grids Dialog

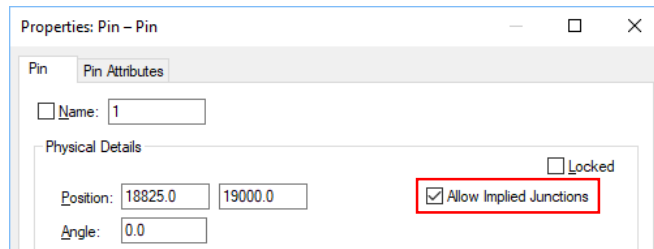
The nested check boxes for **Display** settings on the **Grids** dialog have been replaced with a single combo box enabling you to make the choices required.



Display Implied Junctions on Individual Pins on a Symbol

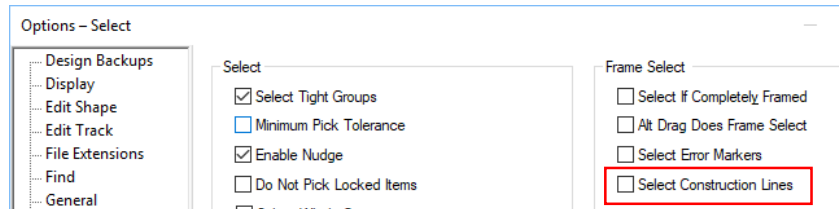
You can now turn implied junctions off on an individual **Symbol Pin** from within the **Symbol Editor**. For example, a signal reference, but this can be used on any Symbol.

To do this, select the pin in the **Symbol Editor** and choose **Properties** and uncheck the **Allow Implied Junctions** so that they are not allowed in the Schematic design.



Frame Selection of Construction Lines

There is a new check box in the **Options** dialog, **Select** and under **Frame Select** to allow selecting **Construction Lines** when using **Frame Select**. This enables you to select and delete multiple Construction Lines in one go.

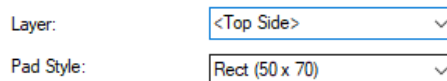


The **Selection Mask** has also been changed to ignore this switch and always select multiple construction lines if they are enabled in the mask.

Insert SMD Pad in Footprint Editor

From within the footprint Editor, you can now select the **Insert SMD Pad** command from the menu. As the name suggests, this enables you to add a pad that will be surface mounted and will be placed on the default **Layer** specified.

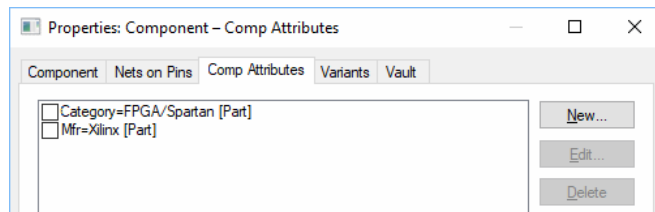
The default SM pad selected will be taken from the **Technology** dialog, **Design Settings, Defaults** page under **SMD Pad**.



Properties

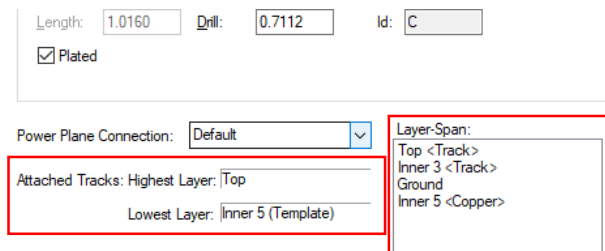
Component Attributes

Component Attributes in the **Component Properties** dialog now include a suffix of [Part] or [Symbol] if the attribute is not actually specified on the Component but derived from one of those sources. If a local source, this is left blank. This is particularly useful if you wish to inspect a Component in the design and view the source of a STEP model (local Component, Part Library or Symbol [Footprint] library).



Via Attached Tracks & Templates/Planes

The **Attached Tracks** section of a **Via's Properties** now reports whether it is attached to a template/plane connection on its highest/lowest layer. There is also an adjacent list box reporting all layers the Via(s) span.

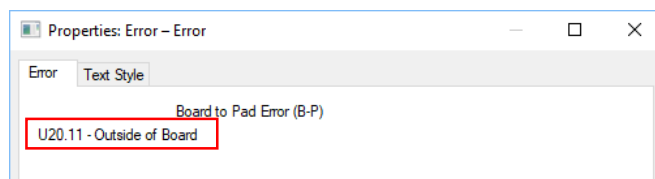


Error Marker Information

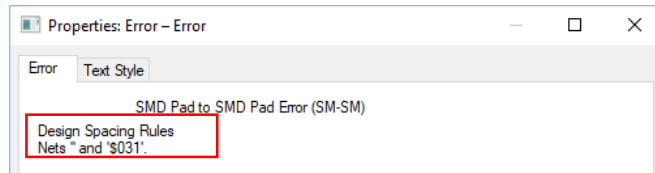
For a selected **error marker**, when using **Properties** of that error, it now reports the rule that was the reason for the error. This information is also reported in the **DRC report**. Note, this information is not available for all error marker codes.

This field can also be reported in the **Report Maker** using the **Rule Name**. Note, this was available in version 10.0, but was almost always blank.

Two examples of **error marker**, **Properties** information are shown below:

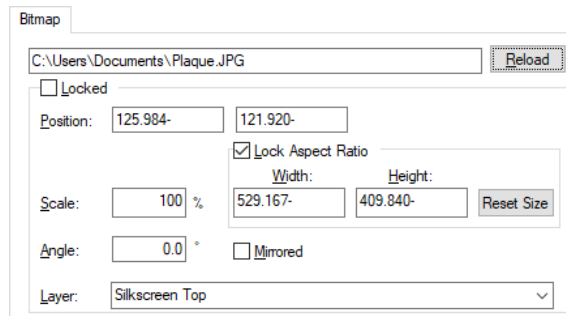


This example shows a more general rule where the nets are also shown.



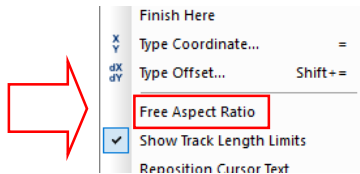
Lock Aspect Ratio and Scale for a Bitmap

The ability to lock the **Aspect Ratio** and **Scale** has been added in **Bitmap Properties**.



Use the **Scale** box to type in the scale of the bitmap. Initially, the scale will be proportional from the bitmap's original scale. Once the Properties dialog has been exited, on running again, the Scale will now show 100%. Changing the scale value will change the image proportional to its current scale.

The aspect ratio is maintained when moving a corner of the bitmap. You can unlock the aspect ratio by selecting the **Free Aspect Ratio** check box from context menu.



Lock Aspect Ratio When Dragging a Bitmap Edge

When dragging an 'edge' (rather than a corner) on a bitmap, the bitmap will now retain its aspect ratio. If you wish to remove the aspect ratio while dragging an edge, deselect it first in the **Properties** dialog using the **Lock Aspect Ratio** option (see above).

Attachment Information for Dimension & Text Callout

The **Properties** dialogs for **Dimensions** and **Text Callouts** now show the attachment information for what they are attached to.

Linear or Angular Dimension

Attachment Information

Attach Start: Copper Segment

Attach End: Copper Segment

Radial Dimension

Attachment Information

Attached To: Doc Shape Segment

Layer Stack Appearance

You can now change and enhance the appearance of an inserted **Layer Stack** Doc Shape in the design.

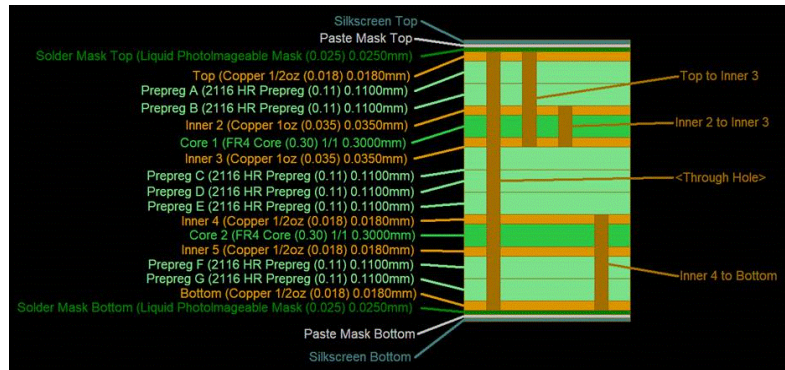
Layer Stack symbols use callouts, these can have a fill colour. By selecting the **text labels** (the callout) and using **Text Callout Properties**, you can the callout **Colour** and **Fill** colour. The box in the layer stack diagram that the label is pointing to will be drawn in the callout fill colour. You can also change the **Text Style** and **Line Style** used.

Box Attach At: Sides Only

Own Colour

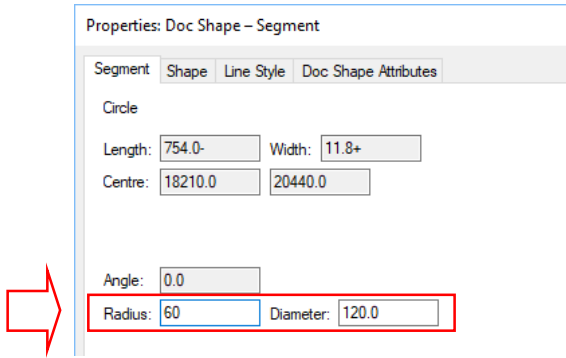
Text Box Fill Colour

Attachment Information



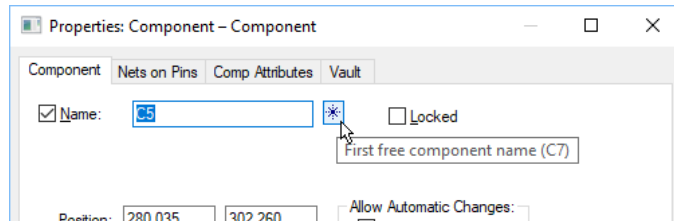
Edit Radius/Diameter in Properties

The **Radius** and **Diameter** for **Circles** and **Arcs** can now be edited using the **Properties** dialog of the selected shape.



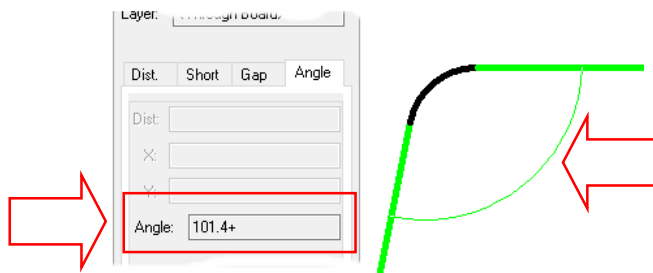
Next Available Name in Component Properties

There is a new button on the **Component Properties** next to the component **Name** field. When selected, this will give you the first free component name. If for example, you have C5 in Properties and C6 is already used, C7 will be the next available in the sequence that is free.



Measure Tool - Angle tab

When using the **Measure** tool, there is now the option to show the Angle between the sides of selected items. This is displayed on the **Angle** tab. A curved measure line will be displayed to show which two sides are being measured.



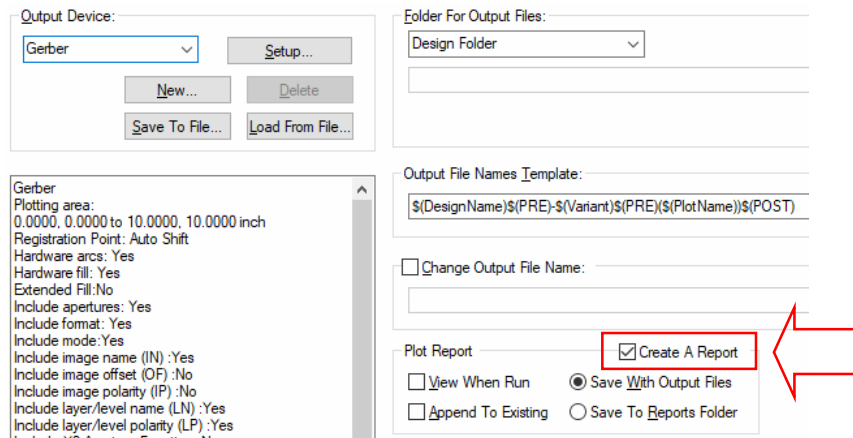
Double Click to Edit Mitre changes

Double click on a mitre will now only enter the **Edit Mitre** mode if it is less than twice the default mitre size in the **Options** dialog and **Edit Track** option.

Changes to CAM Plot Dialog

Ability to Switch Off CAM Plot Reports

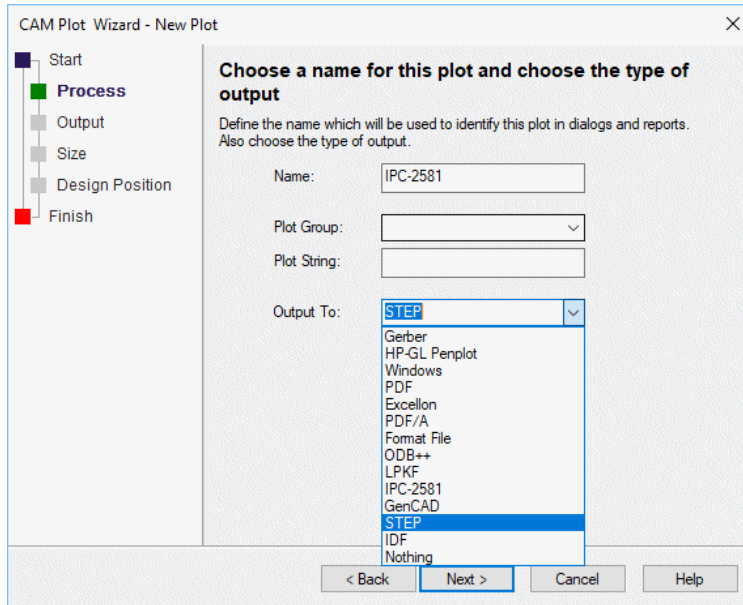
You can now turn off CAM Plot reports using the new switch in the **CAM Plots** dialog, **Plot Settings** and **Create A Report** option.



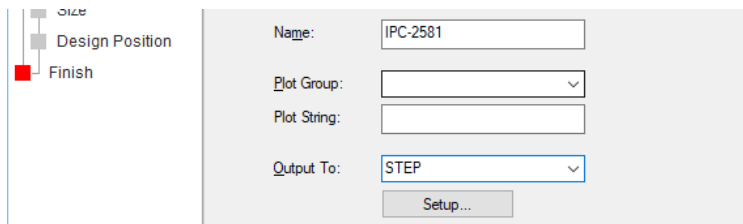
Additional Outputs Types Available Through CAM Plot Dialog

From within the **CAM Plots** dialog, you can now export ODB++, LPKF, IPC-2581, GenCAD, STEP and IDF in addition to the other plot types available. These will be available if have the appropriate license.

Using the **CAM Plot Wizard** and **Process** page, you can access the additional formats using the **Output To:** drop down list.



The **Setup** button enables you to set up the output file. Be aware though, changing the setting from here will also change the setting on the export option itself and not just the plot setting.



The output will be listed in the CAM Plot dialog ready to be run:

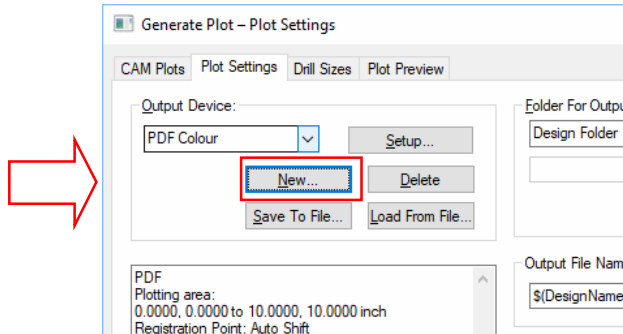
CAM Plots			
Name	Enabled	Device	Process
Top Electrical	<input type="checkbox"/>	Gerber	Layer Top
Bottom Electrical	<input type="checkbox"/>	Gerber	Layer Silkscreen Bottom
Top Silk Screen	<input type="checkbox"/>	Gerber	Layer Silkscreen Top
Bottom Silk Screen	<input type="checkbox"/>	Gerber	Layer Silkscreen Bottom
<Through Hole>	<input type="checkbox"/>	Gerber	Layer Span <Through Hole>
Top All	<input checked="" type="checkbox"/>	PDF	Layer Top
IPC-2581	<input checked="" type="checkbox"/>	IPC-2581	Layer Silkscreen Top

Named Output Devices in CAM Plot Dialog

In the **CAM Plot** dialog and **Plot Settings**, you can now create named output devices. For example, you could have a **PDF Colour** and a **PDF Grey** with different settings. These can then be used for individual plots.

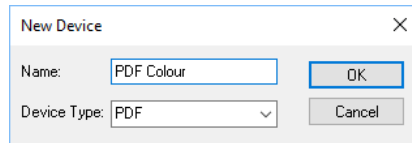
From the **Plot Settings** dialog, you have the **New** button, from where you can create a new named output device based on your current selections. For example, you might make colour selections for a

PDF file and save it as **PDF Colour**. Likewise, you can create another one called **PDF Grey** to use for grey scale PDF printing.

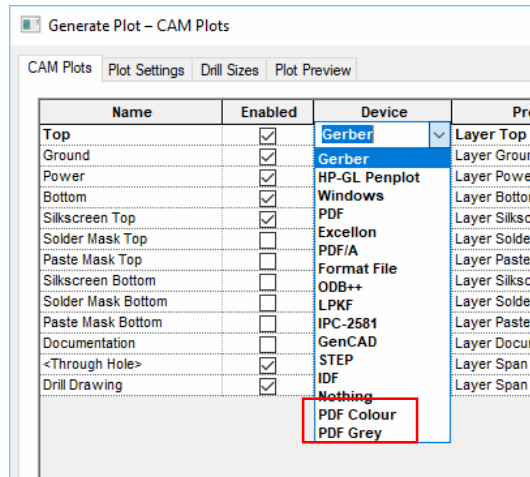


Use the **Delete** button to remove the current user named **Output Device**. The system devices cannot be removed, the Delete button is greyed out if they are selected.

Using the **New** button to create the **name** of the output device and to choose the **Device Type**.

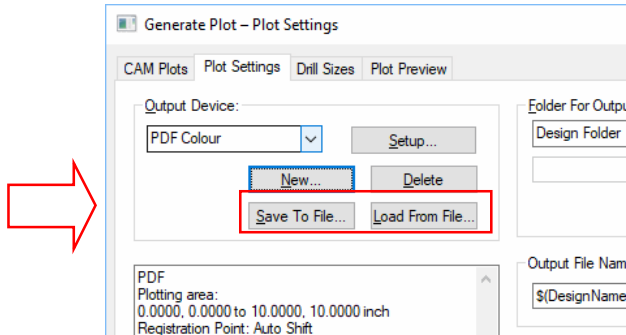


The **Name** will appear in any list of plot devices available



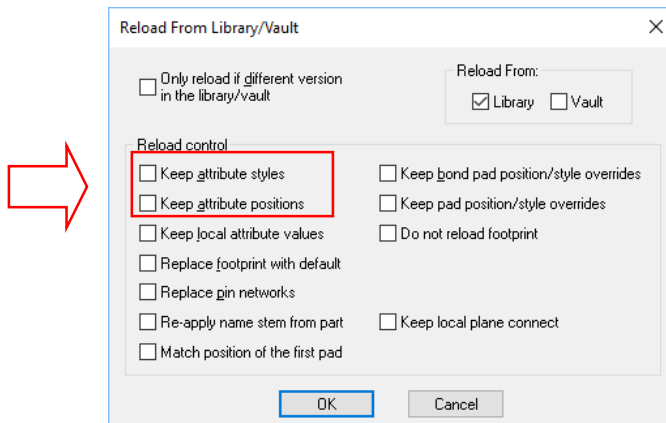
The new devices appear at the bottom of the devices list and are available for selection. You can use these on single or multiple plots as you wish.

You can also use **Save To File** and **Load From File** options to save and load plot files (.plt) if you wish to use them on another machine.



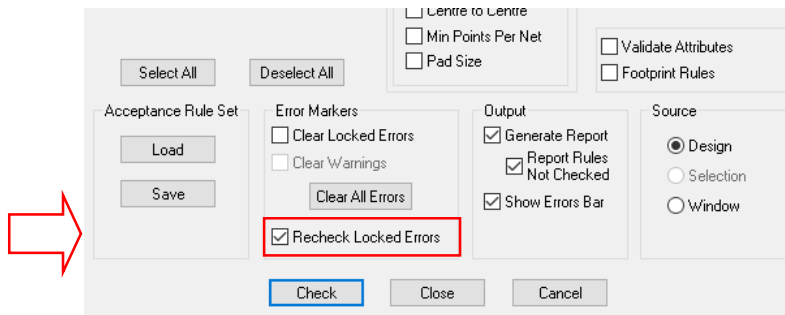
Separate Options in Reload From Library for Styles and Positions

Within the **Reload From Library** dialog, there are now separated entries for **Keep attribute styles** and **Keep attribute positions**. These are available for reloading Parts from the library, Change Part, Alternate Part and Replace Part options.



Recheck Locked Errors

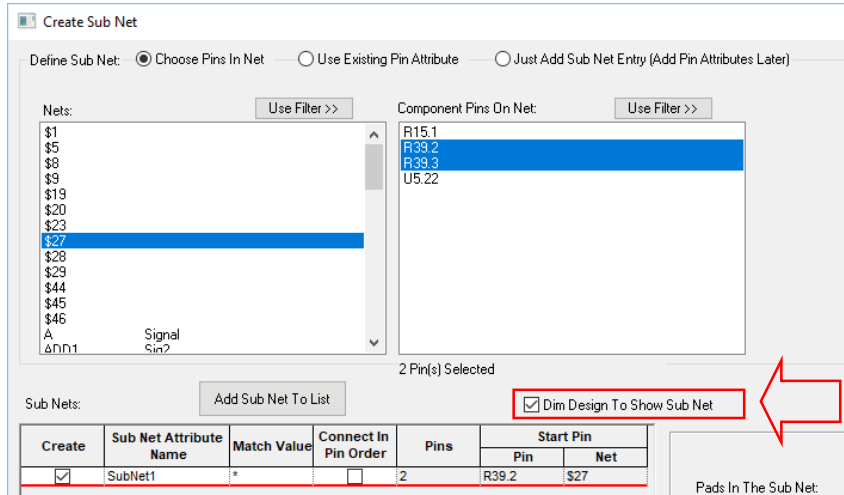
There is a new DRC option to **Recheck Locked Errors**. This will revalidate all the locked errors of the types you are currently checking and remove any that would not now be reported as errors.



Create Sub-Net Highlighting and Dimming

The **Dimming** and **Highlighting** functionality within the **Create Sub Nets** dialog has been simplified.

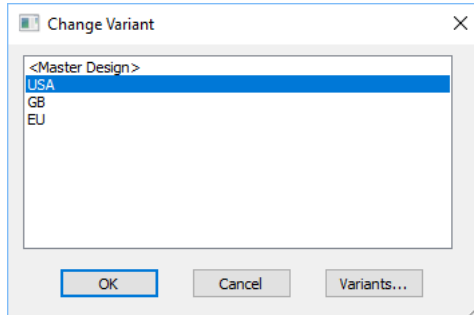
The **Highlight Items In Sub Net** check box has been removed. Highlighting is now always on and you have the option to show it without design dimming by using the existing check box **Dim Design To Show Sub Net**.



Change Variant

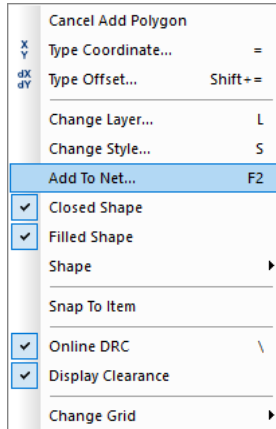
When selecting **Variants** from the active **Status bar**, the variants are now presented to you as a list. Selecting a variant from the list enables you to change the current variant.

The **Variants** button on this dialog takes you to the main **Technology Variants Manager** page from where you can edit the variants.

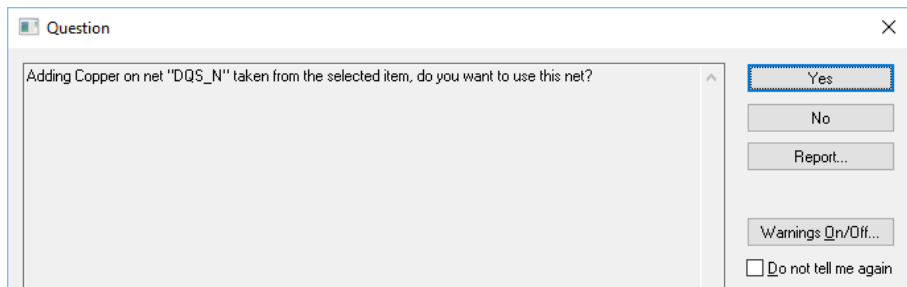


Add Copper & Template To Net During Insert

You can now assign **Copper** and **Template** shapes **to a net** whilst adding them. This can be done either before you start the first point, or whilst adding the shape using **Add to Net** from the context menu or by using the shortcut key **F2**.



For **Copper**, if you have a similar item preselected, the net name will be inherited from this as it does already for **Templates**. This functionality has been changed slightly so that if you preselect a different type of item, a track for example, the net name will be inherited from that item and a warning issued to ask if you want to use that net.

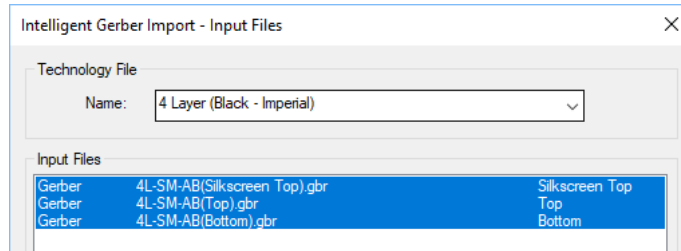


If, whilst adding the shape, if you change your mind about it using a net name, use **Change Net (F2)** from the context menu and clear the name in the dialog.

Intelligent Gerber Import Drag and Drop Files

You can now use drag and drop to deploy files onto the **Intelligent Gerber Import** dialog (in addition to the current use of the **Add** button).

Select your files (Gerber, Excellon, BOM) in the **Windows Explorer** and drag onto the **Intelligent Gerber Import** dialog. As each file is read in, you will see the **File Settings** dialog displayed from where to make your layer mapping choice.



The files are displayed once the layer mapping page has been used.

Report Maker Changes

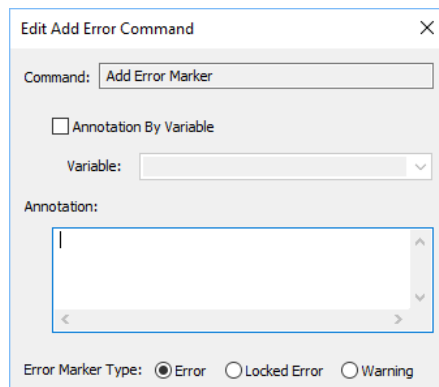
New Commands - Add Error Marker and Delete Error Marker

New commands have been added, **Add Error Marker** and **Delete Error Marker** are now available for a **design**.

Add Error Marker can be used at any time to add user defined error markers of type **UDE** into the design, for example to mark out-of-date Parts or Symbols, or to check the design date is later than one specified.



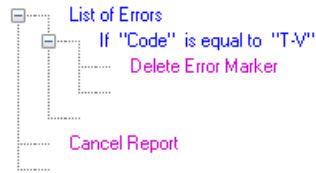
To use this, you will edit the command to define its **Annotation** text and whether it is an **Error**, **Locked** or a **Warning**.



The annotation text can be a multi-line text string or can be defined by the contents of a named variable at run-time. The position will be taken from the current item in the report, or if the current item is the design or does not have a position it will appear at the bottom left corner of the design extents.

Delete Error Marker

Delete Error Marker can be used within a **List of Errors** command to delete any design error marker.



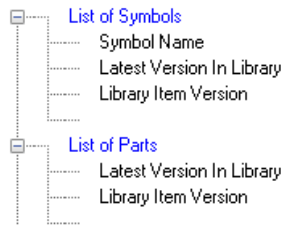
Note, after using the **Delete Error Marker** command, add the **Cancel Report** command to disable an empty report being displayed.

New Report - Out of Date Vault Item Warnings.rff

There is a new report supplied with V10.5 as part of the standard installation called **Out of Date Vault Item Warnings.rff**. As well as reporting out of date Vault items, it also optionally adds warning to the design if the Component is out of date.

New Commands - Library Item Version and Latest Version In Library

New commands, **Library Item Version** and **Latest Version In Library** are now available under **List of Parts** and **List of Symbols** for a design, Parts, Footprints, SCM Symbols, Doc Symbols and Associated Parts.



Library Item Version reports the last saved date and last saved time together as a single version text. The **Latest Version In Library** command reports the latest saved date and time of the item in the library.

New Commands - Latest Revision In Vault and Latest Revision Date In Vault

New commands, **Latest Revision In Vault** and **Latest Revision Date In Vault** are now available for a design, Parts, Footprints, SCMSymbols, Doc Symbols and Associated Parts.



These commands report the latest revision name and date in the vault so that they can be compared with the revision name and date of items in a design. They need access to the vault to work.

Note, the **Date** commands should be copied to variables of type **Number** when used for comparison to avoid wrong text comparisons. The number will be zero if the item was not found.

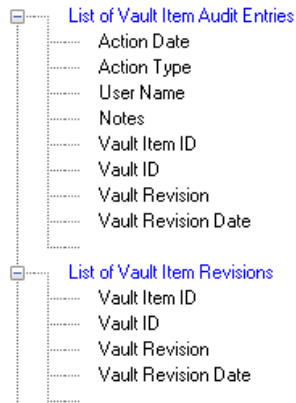
These commands can be used to check a design to see if it has the latest revisions of its Parts etc.

List Commands – List of Vault Item Revisions and List of Vault Item Audit Entries

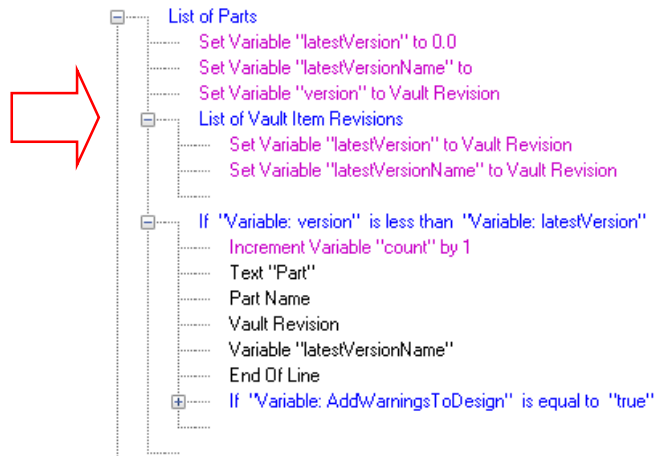
List Commands, **List of Vault Item Audit Entries** and **List of Vault Item Revisions** are now available at the design level. These can be used to insert a version history into the design.

64 Pulsonix Version 10.5 Update Notes

Note: The Vault needs to be available when updated. This can also be used to report which items in the design (Design, Parts, Symbols and Doc Symbols) are not the latest version.

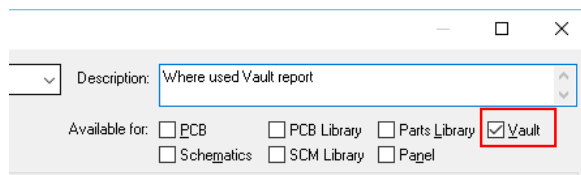


A practical example is presented in the new report - Out of Date Vault Item Warnings.rff:



New Commands - Where Used and Refers To lists from the Vault

New commands have been added to report the **Where Used** and **Refers To** lists from the vault browser. To use these, you must have the Vault check box enabled

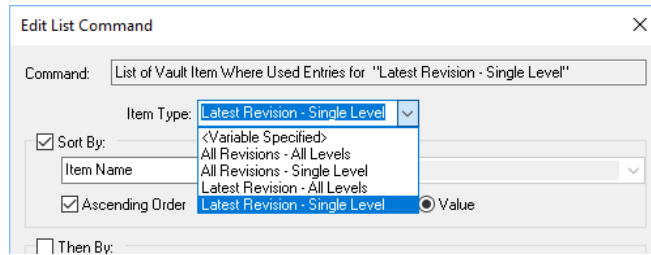


The commands are named **List of Vault Items Refers to Entries** and **List of Vault Item Where Used Entries** can be used within the **List of Vault Items** command.



Both commands can be used for **All Revisions** or the **Latest Revision**, set up by editing the command.

Both commands can be used for **All Levels** or the **Single Level**, set up by editing the command.



For each entry in the two lists you can report the usual vault fields and also use **List of Vault Attributes**.

New Commands - report the Current Path of the library and its Name

New commands have been added to report the **Current path of the library, STEP Model or 3D Package** for items in a PCB or Schematic design. These commands can be used from **List of Components** for example.

The new commands available are:

Current Part Library Name – use on design parts, component groups, components or associated parts.

Current Part Library Path – use on same.

Current Symbol Library Name – use on design SCM Symbols, Doc Symbols, Footprints, Gates and PCB Components.

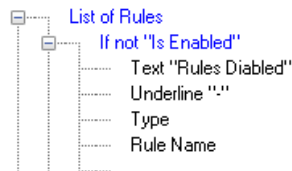
Current Symbol Library Path – use on same.

Current STEP Model Path – use on design Parts, Footprints and PCB Components and also on Footprint library items.

Current 3D Package Path – use on same.

New Command - Is Enabled in List of Rules

New report maker commands support the new **Enabled** check box status of a rule available in the **Technology** dialog - **Is Enabled** is available from within **List of Rules** command on a design. For example, this might be used to report a list of all the rules not enabled using the If not “Is Enabled” command.



The report maker command **Notes** can now also be used when reporting rules.

New Commands - Output Min & Max Spacing Values in List of Rules

Two new commands have been added to write out **Minimum Spacing Value** and **Maximum Spacing Value** for use within a **List of Rules** command. These were added in a late service patch of V10.0. They are only relevant if the **List Rule** type is a **Match Pair Spacing Rule** and they output the min and max values from the Match Pair spacing grid (not including text and board spacings).



New Feature for Variable Command – Convert Text In Value

A new feature for the **Variable** command has been added. The **Convert Text In Value** action allows you to convert the variable value using a specified method.

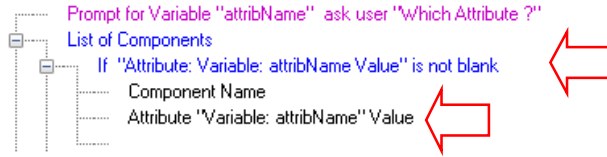
The screenshot shows the 'Edit Variable Command' dialog box. The 'Command' field contains 'Set Variable "version" to Vault Revision'. The 'Variable Name' is 'Name'. The 'Variable Type' is 'Text'. Under 'Action', the 'Convert Text In Value' option is selected and highlighted with a red box. A red arrow points to this option. The 'Action Details' section shows 'Field Contents' set to 'Attribute' and 'Attribute' set to 'MRP NoN'. The 'Use' section has 'Value' selected.

Methods available are **Convert To Upper Case** and **Convert To Lower Case** to change all characters in the text to the chosen case, **Remove Whitespace At Start** and **Remove Whitespace At End** to remove all spaces, tabs and newline characters from the start or end of the text.

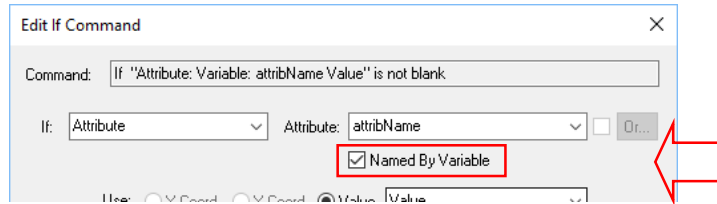
Note: this command was also back fitted to V10.0

Define Attribute Name to Variable

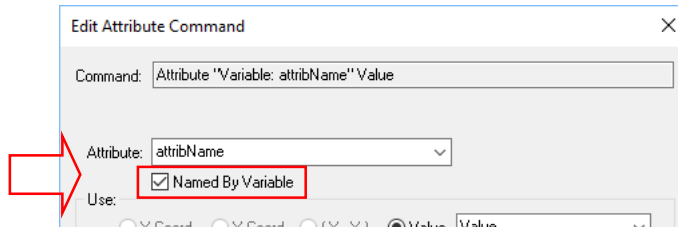
Report Maker is now able to define the attribute name in an attribute command using a variable.



Using the **IF** statement, you can state that the **Attribute** value is **Named By** a **Variable** and the **Variable Name** used as the **attribute** value.



When the attribute is reported, the attribute value is passed the Variable, in this instance, attribName. This is available by checking the **Named By Variable** box. This attribute to be used has already been declared by user input at the start of the command sequence using a user Prompt.

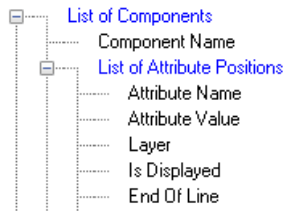


New Commands - List of Attribute Positions

The **List of Attribute Positions** command is used for reporting information about each attribute position on the design, page, component, or on any item that can have attribute positions. For items to be considered for reporting, they must have a physical attribute position defined.

For example, you may wish to report a list of glue spot positions in a PCB design. Each component has a **Glue Spot** attribute with many attribute positions defining where the glue spots are required for manufacture.

Is Displayed – Used within **List of Attribute Positions** and will report True or False. This could be used for example, to write out a list of all the attribute positions that aren't displayed but which need to be. Perhaps they are shown on an Assembly drawing but if not displayed, won't be printed to PDF.



Changed Command - List of Star Points

Updated Star Points.rff Format File

List of Star Points – Used for reporting information about each Star Point in the Design, Page, or Net.

Previous to V10.5, all PCB Star Points were Doc Symbols of type **Star Point**, but now in a PCB, Component Pads, Mounting Holes and Vias can be Star Points.

The **Star Points.rff** format file used within the **Report Maker** has been modified to use this new feature.

New commands available:

Star Point Name – Used within List of Star Points, Pins, Vias, Mounting Holes or Doc Symbols

Is Star Point – Used within List of, Pins, Vias, Mounting Holes or Doc Symbols.

Extra Sort Category on each List of Command added

There is now an extra sort category on each List of... Command. This are now four sort fields on the Sort dialog where previously, it was three categories.

Page Name id now available

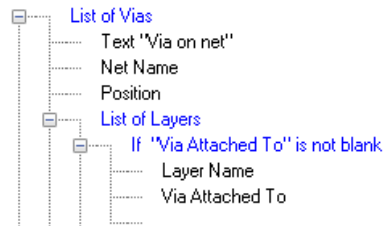
The **Page Name** command is now available from within List of Pins, List of Test Points and List of Star Points. This could be used for example, within a list of components and a List of Pins. The Pins could be on different pages which can then be reported using this new command.



New Command - List of Layers and List of Vias called Via Attached To

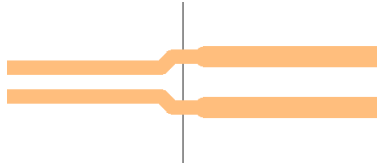
You can now use **List of Layers** within a **List of Vias** command to report all electrical layers used by each via.

In this case, a new **List of Layers** command called **Via Attached To** is available to report the item that the current via is attached to on the current layer. This is reported in the same way as in the **Via Properties** dialog, e.g. <Track>.



Editing Differential Pairs over Area Boundaries

When you adding a **Differential Pair** and crossing an **Area** boundary, if the inside of the area has different track styles or differential pair gaps defined, the pairing is automatically stopped on the near side and started again on the far side. Two transition tracks are added over the area 'border' joining the ends of the paired sections together.



If you have entered the area in error or decide to not enter the area, use the **Backspace** key to remove the area transition tracks and revert to editing the original paired tracks outside of the area.

Changes to Default Supplied Files

Technology Files

All Schematic and PCB Technology files have been updated to reflect the new rule sets in V10.5.

Libraries

The Schematic Doc Symbol library now includes a BUS REFERENCE symbol.