Pulsonix Design System
V14.0 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2025
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 and Windows NT 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: 13/08/25

Pulsonix
20 Miller Court
Severn Drive
Tewkesbury
Glos
GL20 8DN
United Kingdom

Support Phone	+44 (0)1684 296 570
Sales Phone	+44 (0)1684 296 551
Email	sales@pulsonix.com
Web	www.pulsonix.com

Contents

CONTENTS 3
VERSION 14.0 UPDATE SUPPLEMENT 6
Installing the New Version of Pulsonix 6
Customise Feature Installation 6
Licensing 8
Version 14.0 Network Licence Server (NLS) 9
NEW IN VERSION 14.0 10
Tear-off Windows – Use of Dock Bars 10
Customise - Commands Filter 10
Synchronise - Design Changes 10
User Information for a currently Open Design 11
Disable Variant Attributes 11
Component Variant Spreadsheet – 'Reset To Defaults' Cell Colours 12
Library Manager 13
Find Dialog Changes 13
Part Edit / Edit Attributes – Validation String 14
Properties - Edit Rules 14
Property Sheet Search Bar 15
Find in Grids 16
Find – Find Nets In Busses Option 17
Find Net Command – Updated Functionality 18
Fast Category Switching in Parts Browser 18
ERC Changes 19
Treat All Dangling Connections As Unfinished Option 19
Split Net Check Change 20
Alternative Error Marker Symbol 20
Allow Moved Names on Locked Components 20
Changes to Schematic RINF PCB Netlist Export Format 21
Changes to the Select Mode context menu 21
Technology Changes 21
Circular Hatching Style 21
Differential Pairs – View Chain Option 23
Differential Pairs - Report 23
Find from Differential Pairs Dialog 24
Find from Signal Paths Dialog 25
Check Spacing Values Net Pre-selection 25
Same Net Drill Spacing Values for Plated Drills 26
Net Class Level – Name Colour Coding 26
Back Drilling – Inner Start Layer 26
Layer Materials – Density field 27
Load Technology 27
Ignore Design Level Attributes for Symbols option 27
Named Acceptance Rule Sets Listed for Save/Loading Technology Files 28
Copper Pour – Only remove Isolated Islands that are too small 28
Options Dialog Changes 29
Weld Spot Shape 29
Weld Spot Size 30

Simplified Display dialog in Options....30
3D Viewer Changes....30
STEP 3D Viewer Performance Improvements....30
STEP 3D Exploded View Performance Improvements....30
STEP Hierarchy Bar....31
Change Variant in 3D Viewer....31
Flexi Bend Region – Lift-off State....32
Interactive Clipping Plane....33
Type Coord and Type Offset in 3D Viewer....35
Generate Height from STEP for Selected Components....35
3D Settings – Dialog Reorganisation....36
Component Model Selection for 3D Viewer....36
New Area Colours....37
New Display Mode selection....37
Single Entity Merge – Preserve Colours....39
New Measure and Clash Marker Colours....39
Component Keep Out Clash Detection....40
Changes to 3DIgnoreClash STEP Attribute....40
New <STEP Suppress> Attribute....40
STEP Model filter in Position STEP Model dialog....41
Measure option in Position STEP Model....41
Outputs using Direct2D Drawing....41
New command to Rotate Clockwise One Step....42
Apply Vias/Pads – New Options....42
Apply Layout Pattern Apply to Attributes Only....43
Hatch Style Override in Template Properties....43
Track Properties – Display the Layers of Start and End Nodes....44
Show Only Selection Layer....44
Via Properties – Ability to add Via Attributes....44
New Cutout Text option....44
Layers Bar – View Layers option....45
Fillet/Mitre on Corners with an Attached Arc....46
Insert Spiral – Spiral Around Shape....46
Dynamic Snapping when Editing Shapes....47
Construction Lines - Copy With Offset Dialog Change....48
DRC Changes....48
DRC Error Bar – Sub-totals of Errors Found....48
DRC Named Rules Sets (Acceptance)....48
Online DRC – Check Track Layer Option....50
Improved Precision of Silkscreen Overlap DRC Check....51
Apply to Column....51
Odd Angled Track Check....51
Colours - Highlight Odd Angled Tracks / Connections....52
Design Settings – Save default ‘Plot Drilled Out Pad Holes’ status....52
Interactive HTML BOM Option....53
Import Allegro PCB....55
Graphical Design and Symbol Comparison....56
GenCAD Export....60
Neutral Net Names option....60
New options to Include Doc Shapes and Text in Export....60
Vault Changes....60
Using the Vault in V14.0....60
Updated Vault Database (to version 1008)....61
Vault Setup – Visibility of tabs reflects the Sign-in status....61

Vault Setup – Visibility in Options page reflects the Sign-in status 62
Spice Models available in Vault 62
Format Files available in Vault 63
Vault Browser – View Text Vault Files 63
Vault Audit trail – Filter the Audit Trail on Item ID 64
Vault Browser – Synchronise Selected Item 64
Vault Reports – Report Vault Parts without Symbols 64
Recent Notes option in Vault 65
Vault Browser – Folder Search Filter 66
Type Specific Vault Revision Schemes 66
Vault Browser – Folder Searching 67
Vault Attributes in Browser Grid 67
Vault Audit Trail Define CSV Separator For Report 68
Import Vault Users from CSV 68
Vault Browser Shows Design Previews 70
Update Vault Cross-References Progress Dialog 71
Pulsonix Database Connection (PDC) Option 71
Database Check and Update - Select Part dialog 71
Overriding the Default Database Query Timeout 72
Report Maker Changes 73
Major Update to Scripting – Support for Python 3.8 76
Features included for the new Scripting Mechanism in V14 77
Scripting Pad Styles (Legacy Scripting) – New Objects - Plated and ForUseBy 78
Pulsonix Sim – Spice Simulator 78
Using PulsonixSim 78
Library Manager - SPICE Models Page 87

Version 14.0 Update Supplement

Installing the New Version of Pulsonix

It is always recommended you back-up all libraries, designs, Technology Files, Profile Files and Report Maker Format 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. Follow the on-screen commands from the install wizard. The wizard is made up of tabbed pages of a dialog, each one has a different topic and should be reviewed. Some need additional information from you and your choices entered.

By default, a folder name of the current version is offered, it is recommended you use this. There is no advantage or disadvantage in installing Pulsonix over the top of the existing version.

If you prefer, you can uninstall the old version once the new version is running satisfactorily.

Installation Folder

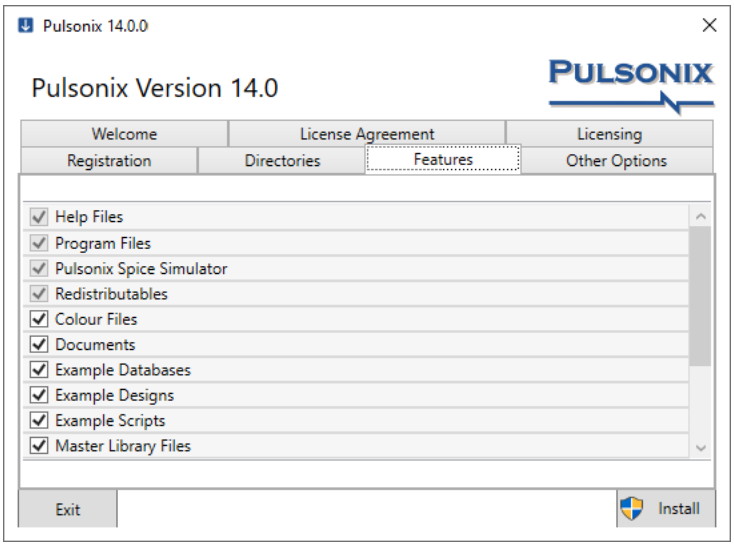
By default, Pulsonix will be installed into the 64-bit programs folder **C:\Program Files\Pulsonix14.0** and not C:\Program Files (X86).

Documentation Installation

The default installation locates all Pulsonix 'documents' (Master Libraries, Technology files etc.) under user\documents\Pulsonix14\

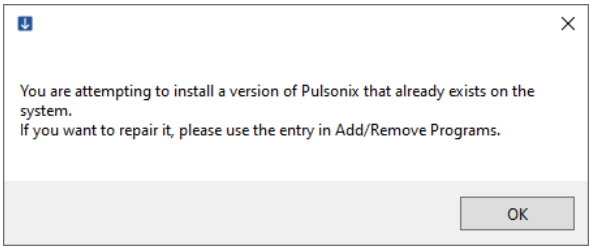
Customise Feature Installation

During a new installation, you can now customise the features that are installed with Pulsonix. Currently, there are 4 required features which cannot be deselected, these are listed at the top of the list. The remainder of the features can be deselected so that they are not installed based on your preference and requirements. You can customise the installed features in **first-time** installation mode and **Repair** mode (selected from Settings on your Windows Start menu).

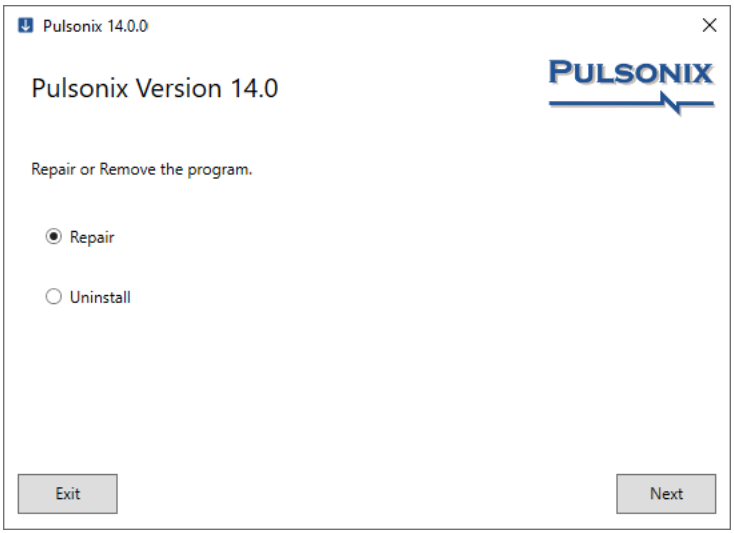


Repair or Uninstall

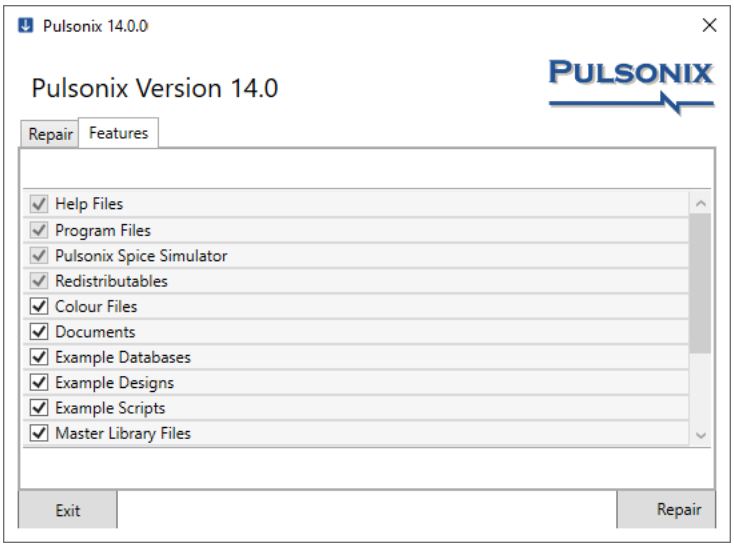
If you already have Pulsonix 14 installed and run the installer again, you will be instructed to run the add/repair option selected from Settings on your Windows Start menu.



From there, you can **Repair** or **Uninstall** the product using the options presented:



When **Next** is pressed, you can now choose the features to install or reinstall if deleted.



Licensing

Version 14.0 requires a new licence if you are a new user or upgrading from any older version of Pulsonix earlier than and including V13.0. The new licence would have been supplied to you under the terms of your maintenance contract or product purchase.

For existing users upgrading from a previous version, it is recommended that you save the new licence 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 licences and make changes to network licensing after the installation has been completed.

Version 14.0 Network Licence Server (NLS)

NLS has been updated for **Version 14.0** but an existing NLS installation of V13.0 will still run V14.0. However, in order to access any new functionality and to take advantage of any issues fixed, you should always install the new version of the NLS program. Older versions (prior to V13.0) will not run V14.0 Pulsonix and cannot be supported.

New In Version 14.0

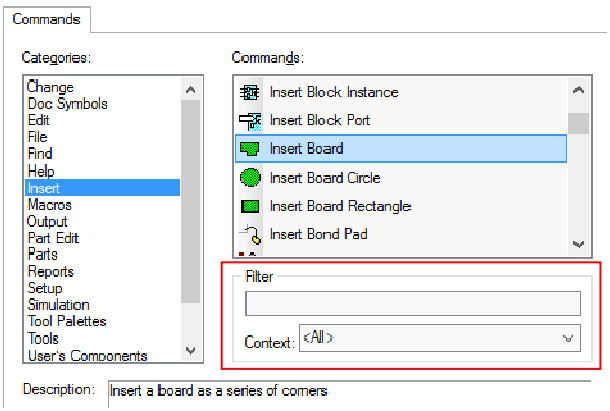
Tear-off Windows – Use of Dock Bars

Torn-off frames now have the ability to use dockable bars, for example, Find, ERC Errors bar etc. However, the Database Connection (PDC) bar is not available on a torn-off frame, but all other dock bars can be used. If the PDC is required, then it can be used in the main Pulsonix window and components directed to the torn-off window in use.

The docking state of each dock bar is copied from the main frame each time a new torn-off frame is created. The docking state is not remembered on restart if it is changed in a torn-off frame.
Dock bars cannot be moved/undocked/floated on a torn-off frame.

Customise - Commands Filter

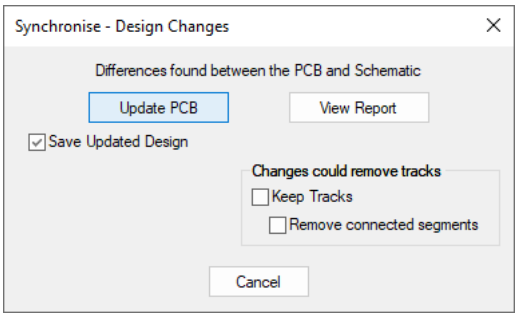
In the **Customise** dialog, a new **Filter** section has been added to the **Commands** page. This allows commands that contain the filter string or are available with the selected design context to be shown. The filtered commands are updated every key press or when the context is changed in the selection box.



Synchronise - Design Changes

When the **Synchronise Design** option is run, the option, **Components will be removed** has been changed to **Changes could remove tracks**, and a sub-option **Remove connected segments** has been added.

The improved functionality can now detect pad changes so you can choose if you want to **Keep Tracks** or just remove the connected segments.

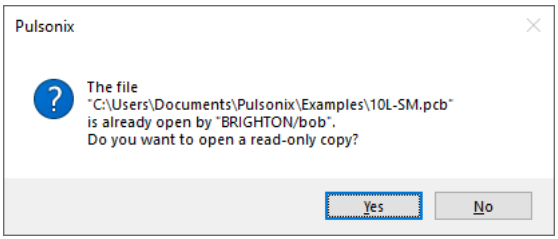


This feature was back-fitted to V13.0.

User Information for a currently Open Design

When a design is opened with write access, a lock file is created containing some details about the user who had opened the design. When attempting to open the same design in Version 14, the lock file is read and the computer name and user name of the person who has the design open is reported as a warning message.

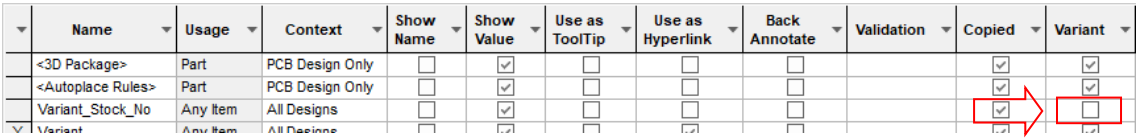
This only applies to Schematic and PCB designs.

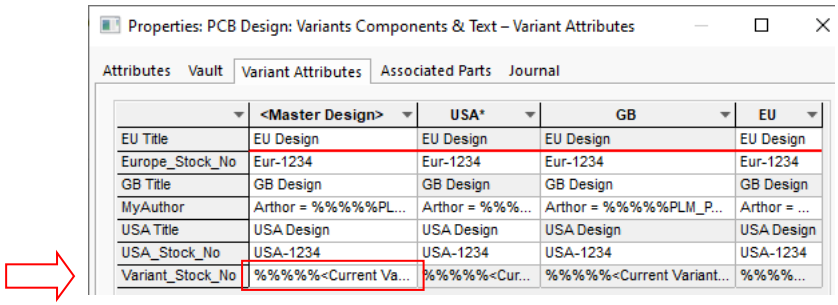


A note about lock files – if a lock file remains after closing and the design is not open, the lock file can be safely removed without any detrimental effect on the design it is related to.

Disable Variant Attributes

From within the **Design Properties**, you can now **Edit** the **attribute value** in **the <Master Design>** column even if the **Variants** field is **disabled** in the **Technology Attribute Naming** dialog.



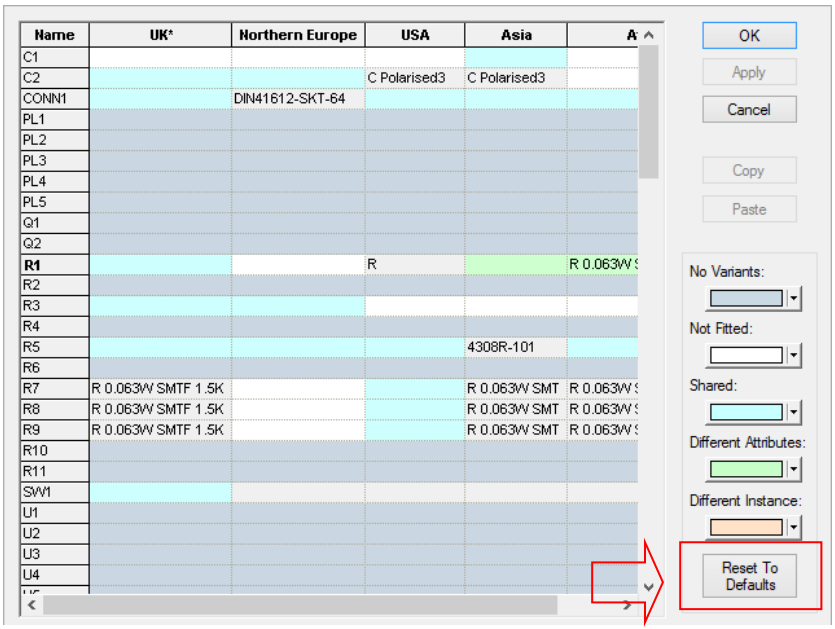


This feature is also available when using the **Inspector Bar**.

This feature was back-fitted to V13.0.

Component Variant Spreadsheet – 'Reset To Defaults' Cell Colours

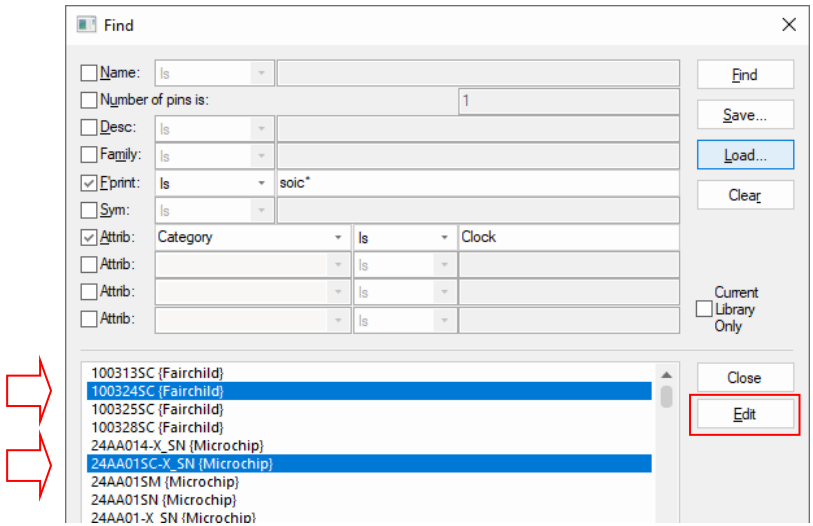
A **Reset to Defaults** button has been added to the **Component Variant Spreadsheet** which, when clicked, sets cell colours back to their original 'factory' values. The original colours will depend on which theme is currently active (for example, if light theme is active, the light set of original cell colours will be used).



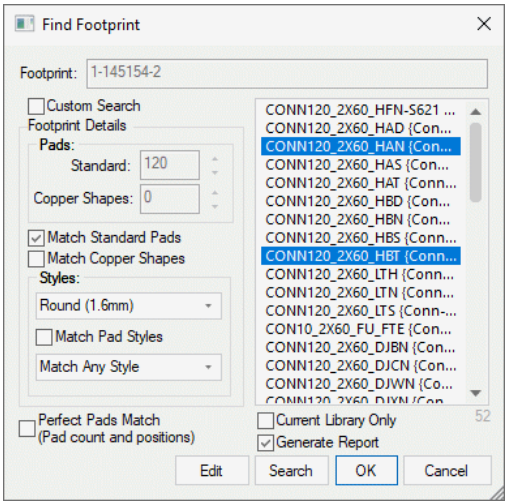
Library Manager

Find Dialog Changes

In the **Library Manager** dialog after using the **Find** option on Parts or PCB Footprints, you can now multi-select items and edit them with the new **Edit** button.



The find PCB Footprints dialog has a new **Edit** button and allows multiple item selection:



Part Edit / Edit Attributes – Validation String

In the **Part Edit** and **Edit Attribute** dialogs, if you select an attribute value that has a validation string (defined in the **Technology** dialog under **Attribute Names**), a drop down list will appear with available attribute values that are based on the validation string (only a simple OR wildcard is supported otherwise the string won't appear in the combo box list).

example 1:

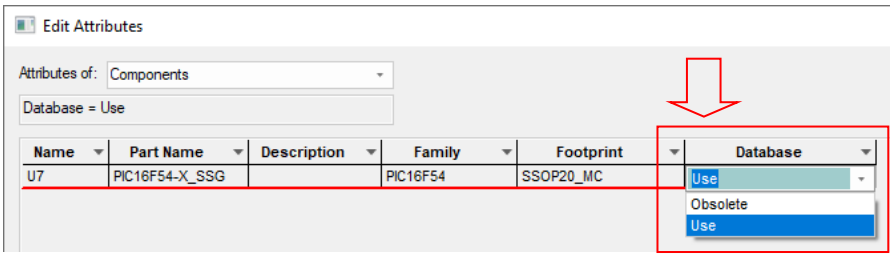
validation string : Test1 | Test2

list : {Test1, Test2}

example 2:

validation string : *Test1 | Test2 ('*' is a wild card)

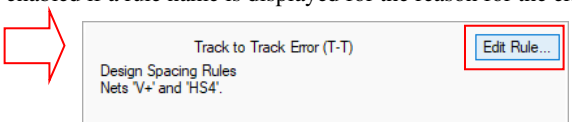
list : {Test2}



Properties - Edit Rules

Error Marker Properties

There is now an **Edit Rule** button available on the **Properties** dialog for an **error marker**. It is enabled if a rule name is displayed for the reason for the error. Not all errors have a rule displayed.

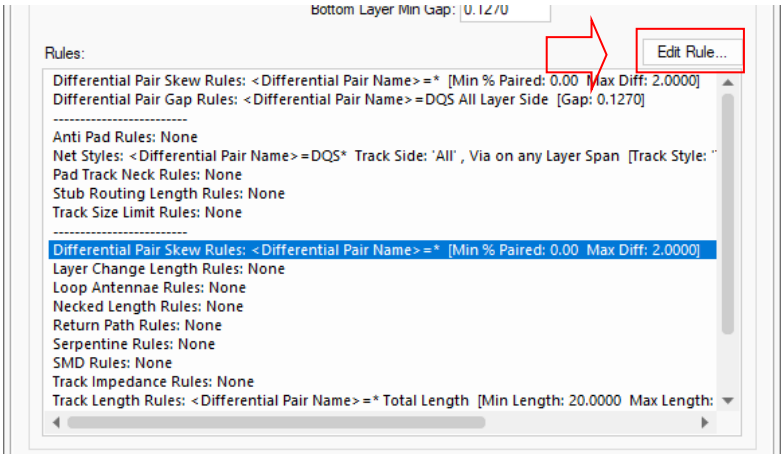


Pressing the button will **apply** the Property page, quit the dialog and run up the corresponding rule page in the Technology, with the appropriate rule selected.

Property Pages with Relevant Rules

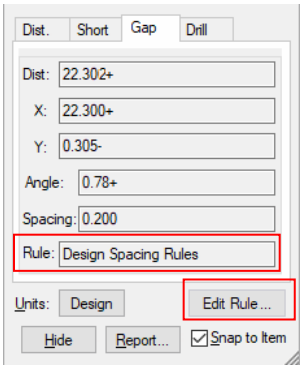
This functionality has also been provided for **Property** pages that list relevant rules. For example, **Differential Pair**, **Net** and **Area**.

To use this, select a rule in the list and Edit Rule will be enabled.



Edit Rule in Measure tool

The **Edit Rule** Button is also available in the **Measure** dialog.

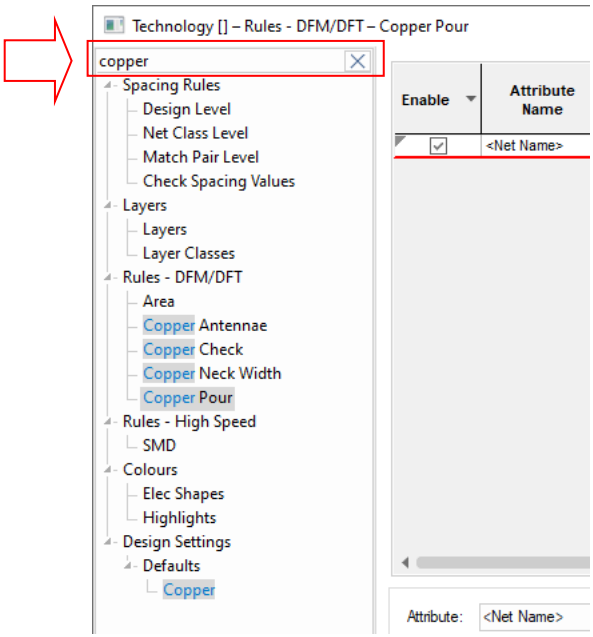


This is enabled if a rule is displayed in the Gap page of the dialog showing the rule responsible for the displayed spacing. Again, pressing the button will quit the measure dialog and run up the correct rule page in the technology, with the correct rule selected.

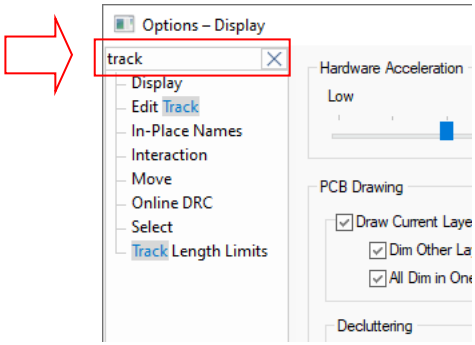
Property Sheet Search Bar

The **Property Sheet Search Bar** is enabled on property sheets with a tree control, within the **Technology** and **Options dialogs**.

By typing into the search bar, you will be presented with results that match the words. Selecting a topic presented will direct you to that page.



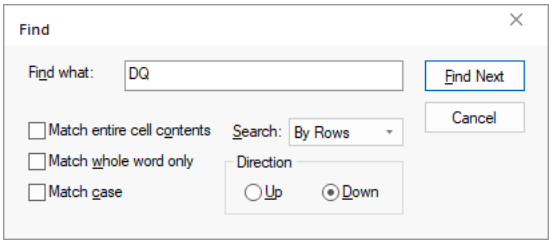
The **Options** dialog also has the search facility:



Find in Grids

From with any dialog grid, such as **Technology** and **Nets**, you can now use the standard **find** command **Ctrl-F** when the grid has focus.

A dialog is displayed from where you can type in the string you wish to find.



Settings are saved when a find is performed, by pressing the ‘Find Next’ button.

These saved settings will be used when the find window is next opened.

The position of the find window is saved when it is closed so it will open in the same location.

There are options that can be changed in the find window that will affect if and how a match is found in the grid:

Find what – This is the string that will be searched for in the grid

Match entire cell contents – Find will return a match only if the find string matches the entire contents of a cell

Match whole word only – Find will return a match only if the find string matches whole words in the contents of a cell. Note: This option will have no effect if 'Match entire cell contents' is checked

Match case – Makes the find case-sensitive

Search

By Rows – Specifies the search order by rows. The whole of a row will be searched until moving on to the next row

By Columns - Specifies the search order by columns. The whole of a column will be searched until moving on to the next column

Direction – The direction options shown are determined by the selected **Search** option

By Rows

Up – The search direction will be from right to left for each visible row from bottom to top

Down – The search direction will be from left to right for each visible row from top to bottom

By Columns

Left – The search direction will be from bottom to top for each visible column from right to left

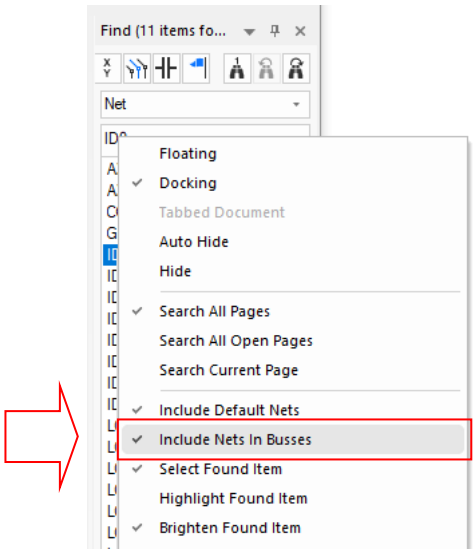
Right – The search direction will be from top to bottom for each visible column from left to right

Find – Find Nets In Busses Option

With a Schematic design, within the **Find** dialog and **Nets**, there is now a new **Find Nets In Busses** option.

If **checked** it will first find the subnets of the chosen net, and then find each bus that contains the chosen net. If **unchecked**, only the subnets will be found and not the actual bus.

When finding nets in a schematic design the Find Bar context menu will also contain the this option.



Find Net Command – Updated Functionality

The **Find Net** command now finds the net of a selected item in the design.

The existing **Find Net** command is available from the **Edit** Menu. It is now also available from the context menu in the main window when an item on a single net is selected.

Using this feature

When selected, it first switches to the Nets find category (as it used to), and then finds the selected net if it is in the find bar list. So if you are using a Nets filter and the net does not match it then it will not be found.

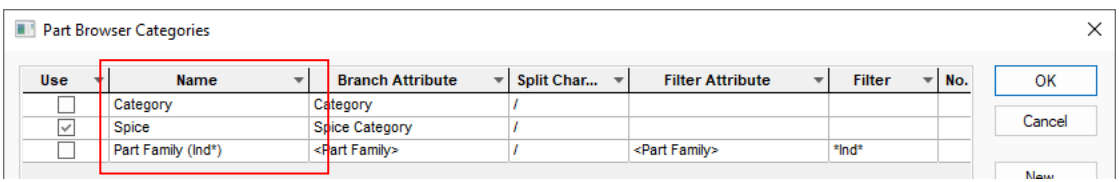
In the PCB or Footprint editors the whole net will be found and displayed using the current Find options.

In the Schematic editor the section of the net containing the selected item will be found and displayed using the current Find options. You can then use the **Find Net Item** and **Find Previous Item** commands (or Find Bar buttons) to find the next section of the net.

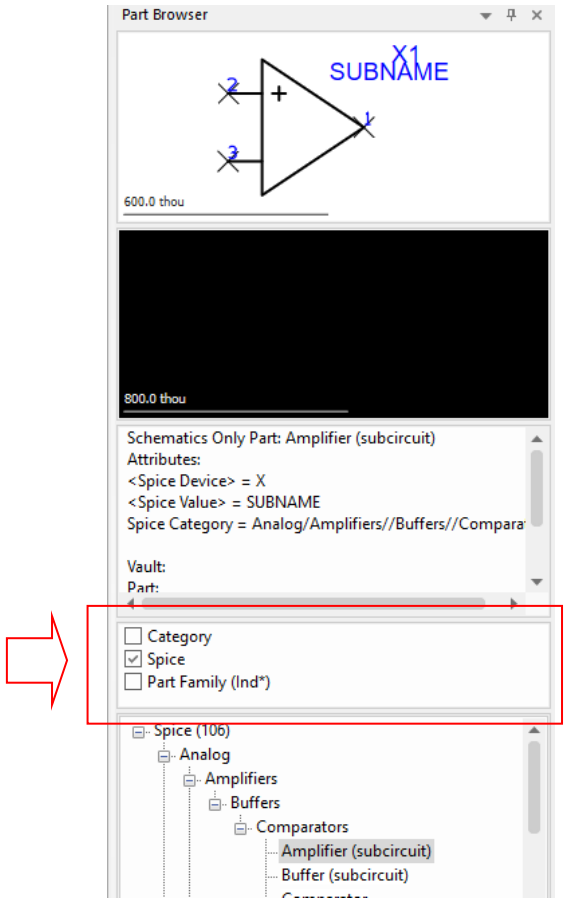
Fast Category Switching in Parts Browser

The **Parts Browser** has a new pane that can be used to change the **Part Category** for Parts displayed in the tree.

Where you have multiple named **Categories**, you can easily and quickly switch these within the Parts Browser window by selecting them by name:



The **Part Browser** then displays the **named categories** ready for selection:

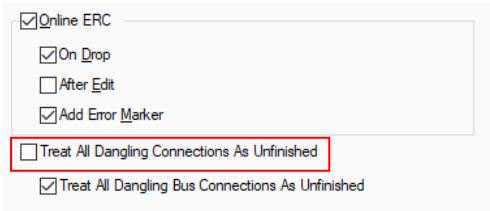


ERC Changes

Treat All Dangling Connections As Unfinished Option

There is a new option in the **Online ERC Options** page, **Treat All Dangling Connections As Unfinished.** When selected, this option causes a connection to be marked as unfinished (i.e. drawn in the unfinished colour) if the connection ends on nothing at one of its ends, even if it is labelled with the Net Name. The default for this switch is unchecked.

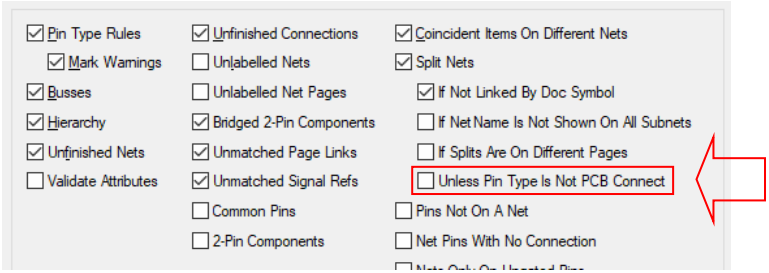
This was added for users who want all dangling Schematic connections to be shown as unfinished. Normally, the presence of a Net Name or Signal Reference means it is not unfinished.



This feature was back-fitted to V13.0.

Split Net Check Change

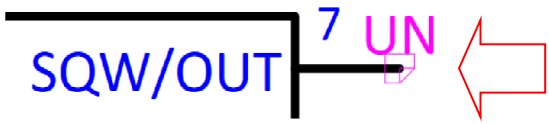
For the ERC split net check, the **Unless ungated pin is PCB No Connect** wording has been changed to **Unless Pin Type is not PCB Connect** to reflect the actual functionality being performed.



This feature was back-fitted to V13.0.

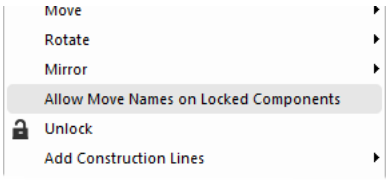
Alternative Error Marker Symbol

An **Error Marker** will now show an alternative symbol for Errors that have **Notes** assigned to them. Instead of the normal origin style symbol, a document style symbol is displayed. This does not affect an Error Marker that shows a line between two items rather than a symbol. This is available for both Schematic and PCB designs.



Allow Moved Names on Locked Components

The select mode context menu items **Allow move of Name attribute on Locked Items** and **Don't Allow move of Name attribute on Locked Items** have been replaced by a checked menu item **Allow Move Names on Locked Components**.



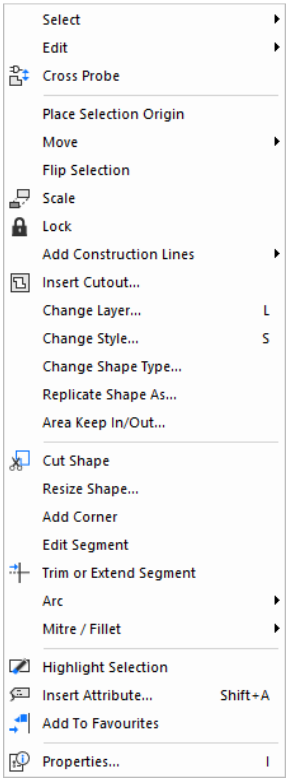
Changes to Schematic RINF PCB Netlist Export Format

The RINF netlist format exported from a schematic design has been changed to include all component attributes in the .COM_ATT lines. The attribute names must be the **All Designs** context to be included in the export.

This feature was back-fitted to V13.0.

Changes to the Select Mode context menu

In order to rationalise the length of the **Select** context menu, there have been some additions to the main menu with some of the options being grouped into sub-options. This change affects options such as Select Component, Select Track, Select Shape etc.



Added new Move sub menu
Added new Rotate sub menu
Added new Mirror sub menu
Added new Mitre / Fillet sub menu
Moved Enable Selection Mask option onto Select sub menu

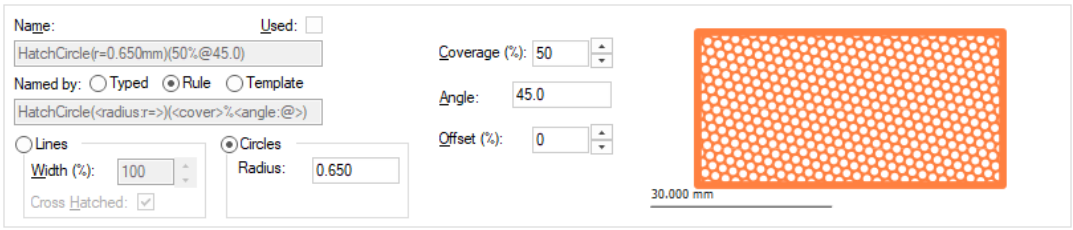
Technology Changes

Circular Hatching Style

A new **Hatching Style** has been added to the **Technology** dialog.

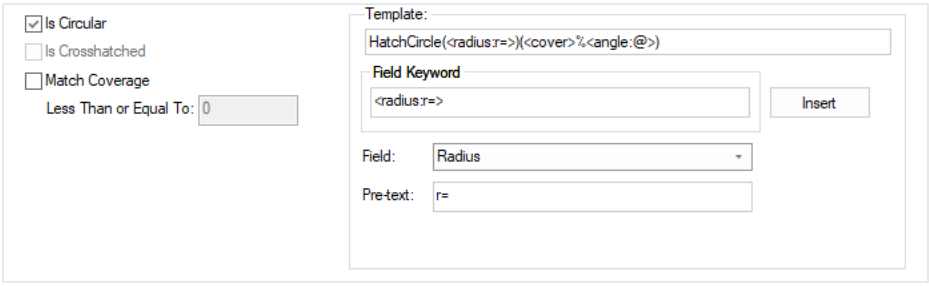
In addition to lines, you can now opt to draw the shape filled with **circular** cutouts.

Additional controls have been added to the **Hatch Style** dialog to define the **Line**, **Width %** and whether it is **Cross Hatched** or not, and the new controls for **Circles**. A circle **Radius** is required for this setting.



Hatch Style Naming Rules

The **Hatch Style Naming Rules** dialog has also bee updated to accommodate this new style.

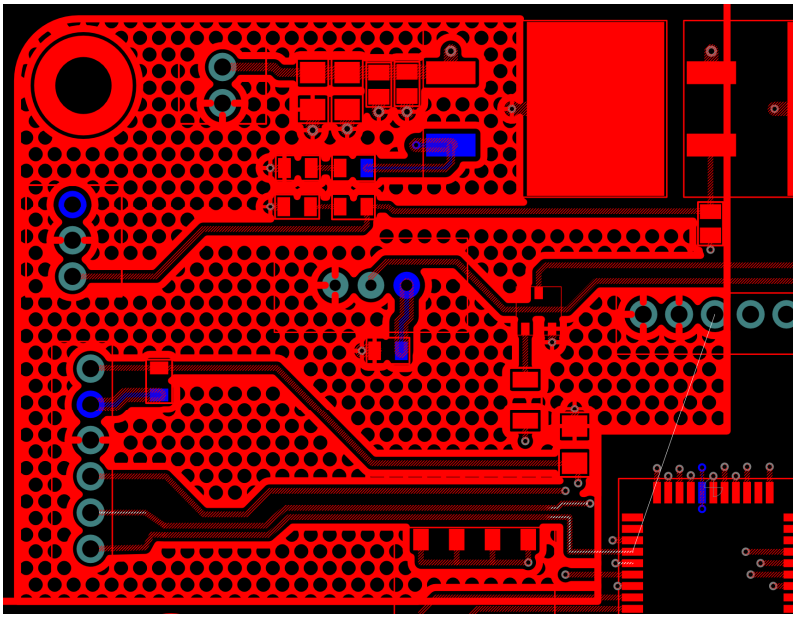


You can set the naming rule to be **Is circular** which checks the styling in the Hatch Style name; it checks if the style is **Lines** or **Circles**.

A new **Field** of **Radius** is available along with a Pre-text of **r=** This reports the radius of the **Circle** defined for the **Hatch Style**.

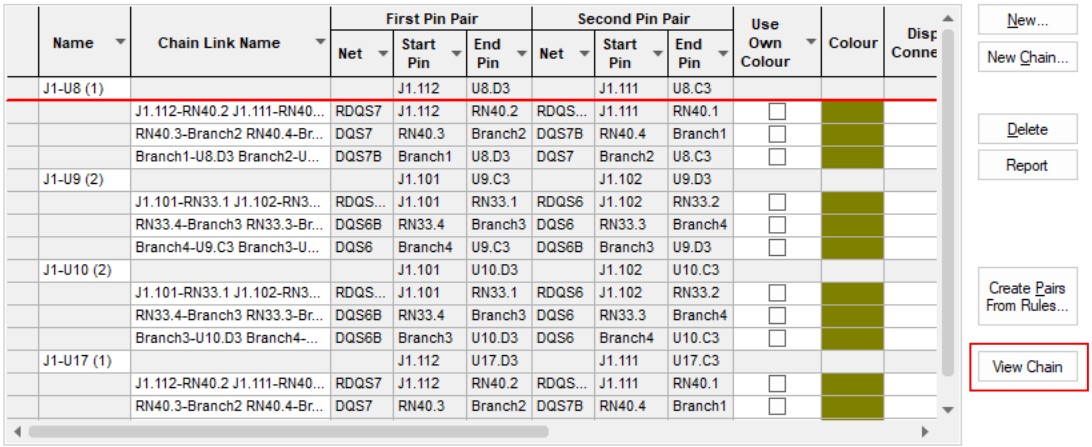
Resultant Copper Hatch Styling

Once the rule is applied to a copper shape, it looks something like this:

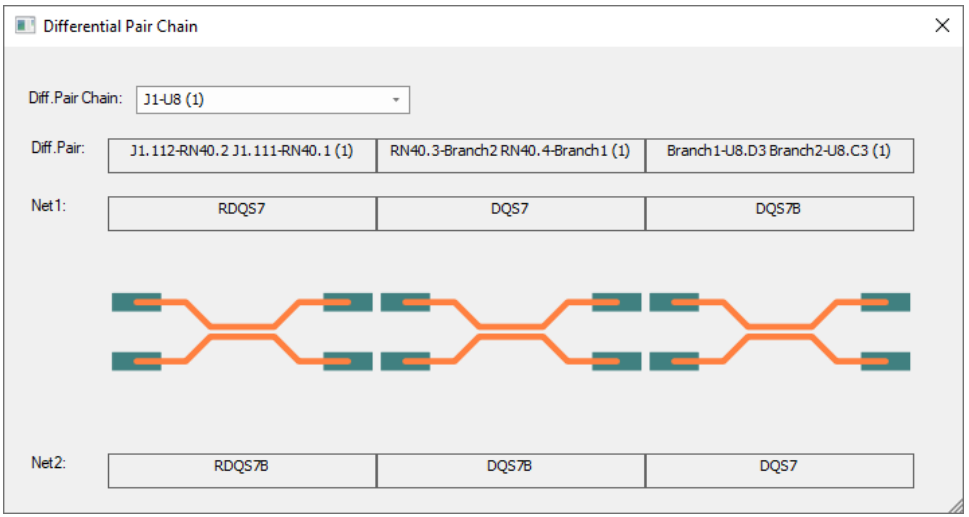


Differential Pairs – View Chain Option

A new **View Chain** button has been added to the **Differential Pairs** page of the **Technology**.



This feature allows you to view a detailed representation of the whole **Differential Pair Chain**.

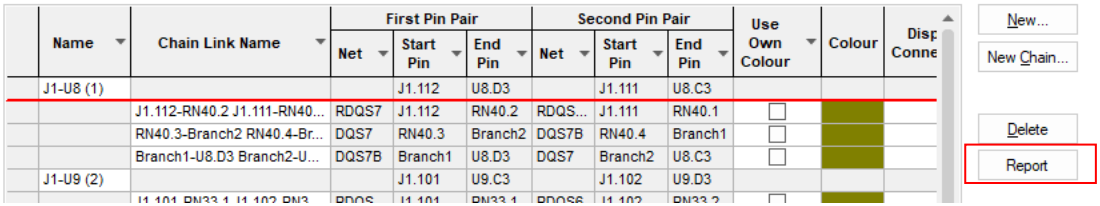


Each Differential Pair in the chain is displayed side by side, providing a comprehensive view of the entire chain structure.

The drop-down box enables you to select and view the different Chains available.

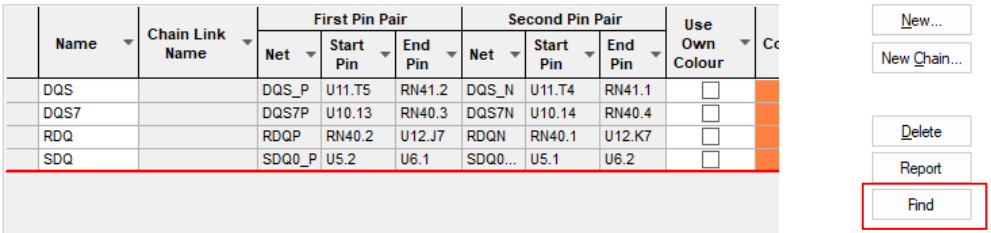
Differential Pairs - Report

A **Report** button has been added to the **Differential Pairs** page of the **Technology** dialog. This reports information about the defined rules of the selected item or multiple selected items.

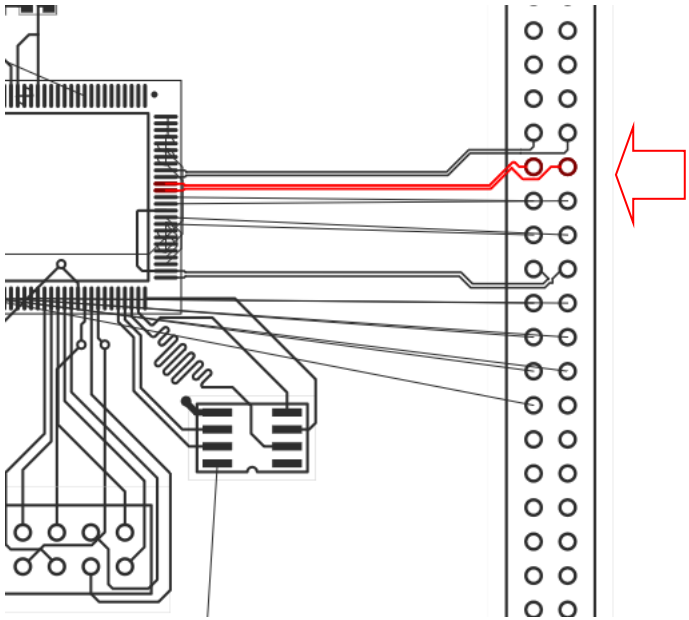


Find from Differential Pairs Dialog

In the **Technology** dialog and **Differential Pairs** page, there is now a new **Find** button. Pressing the find button after selecting a Differential Pair name from the grid will highlight the selected path in the design.



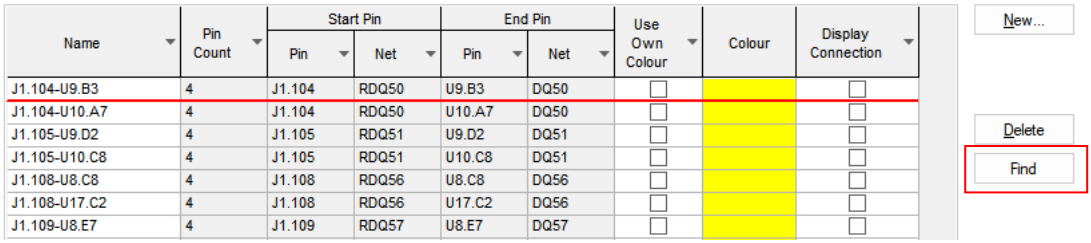
Find will use the standard **Find options** for dimming etc.



This feature was back-fitted to V13.0.

Find from Signal Paths Dialog

In the **Technology** dialog and **Signal Paths** page, there is now a new **Find** button. Pressing the find button after selecting a Signal Path name from the grid will highlight the selected path in the design.

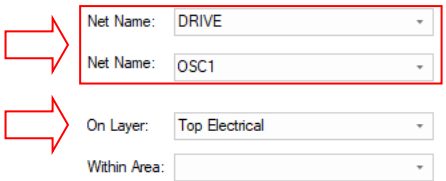


Name	Pin Count	Start Pin		End Pin		Use Own Colour	Colour	Display Connection
		Pin	Net	Pin	Net			
J1.104-U9.B3	4	J1.104	RDQ50	U9.B3	DQ50	☐		☐
J1.104-U10.A7	4	J1.104	RDQ50	U10.A7	DQ50	☐		☐
J1.105-U9.D2	4	J1.105	RDQ51	U9.D2	DQ51	☐		☐
J1.105-U10.C8	4	J1.105	RDQ51	U10.C8	DQ51	☐		☐
J1.108-U8.C8	4	J1.108	RDQ56	U8.C8	DQ56	☐		☐
J1.108-U17.C2	4	J1.108	RDQ56	U17.C2	DQ56	☐		☐
J1.109-U8.E7	4	J1.109	RDQ57	U8.E7	DQ57	☐		☐

This feature was back-fitted to V13.0.

Check Spacing Values Net Pre-selection

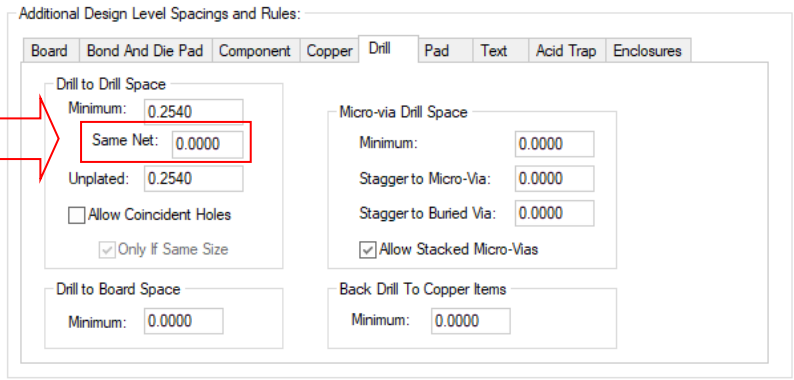
If two nets are selected in the design, when you open the **Technology** and **Spacing Rules** page and **Check Spacing Values**, the two net names will now be pre-selected in the dialog. Also, if both items are on the same layer (in a PCB design), the Layer control is also set in the dialog.



Net - OSC1	Net - DRIVE								
	Track	Pad	SMD Pad	Via	Micro-via	Testpoint	Mounting Hole	Copper	Text
Track	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Pad	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
SMD Pad	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Via	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Micro-via	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Testpoint	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Mounting Hole	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Copper	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Text	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400
Board	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400	0.25400

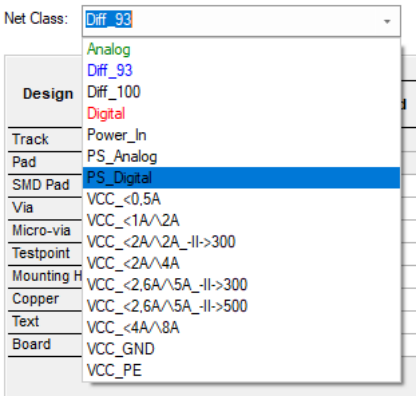
Same Net Drill Spacing Values for Plated Drills

In the **Technology** under the **Design Level Spacing Rules** and **Drill** tab, there is a new control for **Same Net**. This enables you to set the spacing value for two plated drills on the same net.



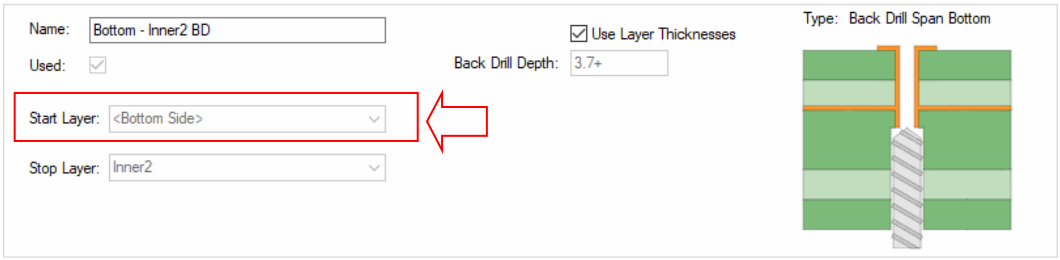
Net Class Level – Name Colour Coding

In the **Technology** under the **Net Class Level Spacing Rules**, Net Class items in the list change colour based on the spacing values. This is similar to the colour coding for Spacing grids - Black for inherited Design Level, Green for minimum spacings, Blue for explicit values and Red for both (green and blue).



Back Drilling – Inner Start Layer

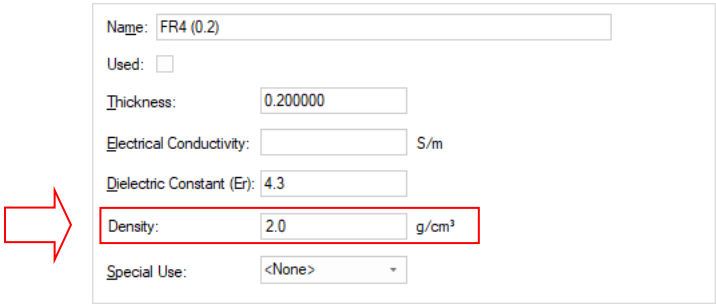
From within the **Technology** dialog and **Back Drill Layer Span** page, the **Start Layer** option for a back drill span has been updated to allow the selection of an **inner electrical** layer (or inner reference layer). This enables back drill spans to be used on vias that do not start on either <Top Side> or <Bottom Side>.



This feature was back-fitted to V13.0.

Layer Materials – Density field

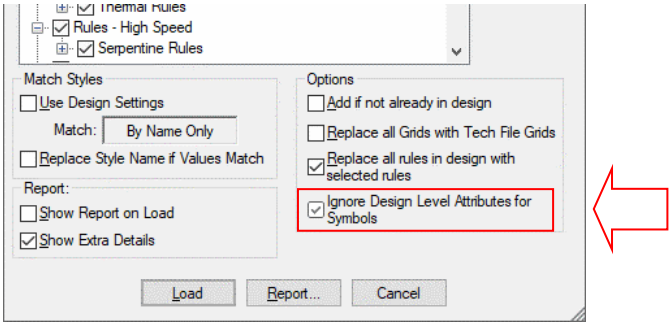
You can now specify the **Density** of a material in the **Materials** dialog. This field will be exported from the **Report Maker** option. The value used is defined in g/cm^3



Load Technology

Ignore Design Level Attributes for Symbols option

There is a new option in **Load Technology** dialog – **Ignore Design Level Attributes for Symbols**.

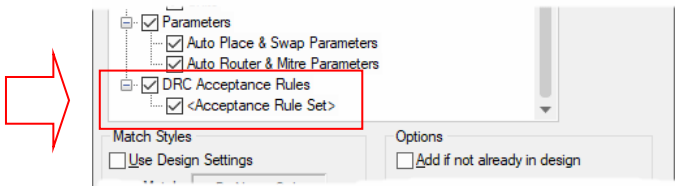


If checked and you are loading the Technology into a Symbol, **Design Level Attributes** will be removed from the list. This enables you to add design level attributes to a Technology file and avoid having them added to a footprint or Symbol when loading the Technology file into them.

This feature was back-fitted to V13.0.

Named Acceptance Rule Sets Listed for Save/Loading Technology Files

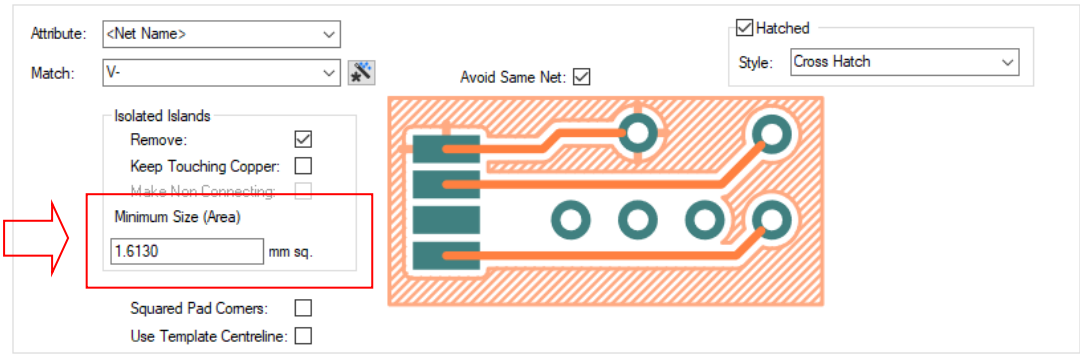
When **Loading** or **Saving Technology Files**, you can now select any **DRC Named Rule Sets** that have been saved.



Copper Pour – Only remove Isolated Islands that are too small

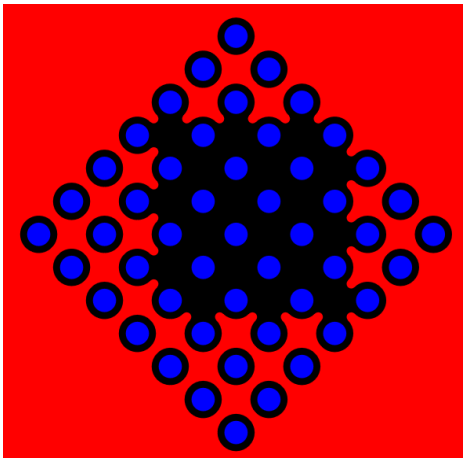
There has been a change to the way touching copper items are handled when considering islands.

From the **Technology** under **Rules – DFM/DFT** and **Copper Pour**, the column **Min Island Size** has been moved to the isolated islands section to reflect the code change.

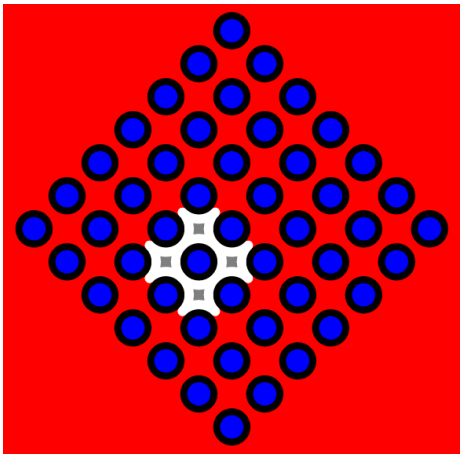


If you have **Keep Touching Copper** checked, then items of copper that touch another item of copper will be counted as connected; this means the minimum island size doesn't apply to them, (the touching items of copper are effectively the same item of copper).

As an example, with **Keep Touching Copper** not selected, it produces this result:



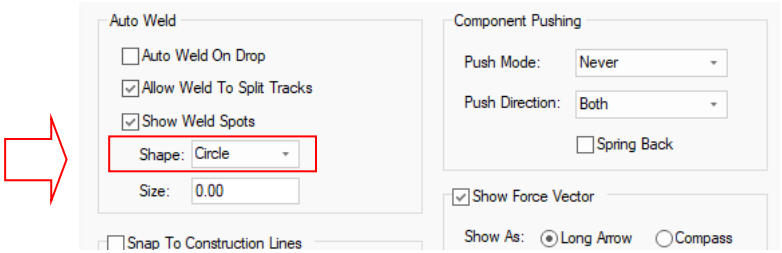
With **Keep Touching Copper** selected, it produces this result. One of the four copper quadrants is selected in the design to highlight the result:



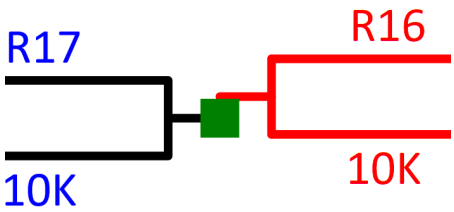
Options Dialog Changes

Weld Spot Shape

In the **Options** dialog under **Move**, using the **Show Weld Spots** check box, you can set the shape of the spots using the drop down list. There are three options available for the shape: Circle, Octagon and Square. The shape is defaulted to circle.

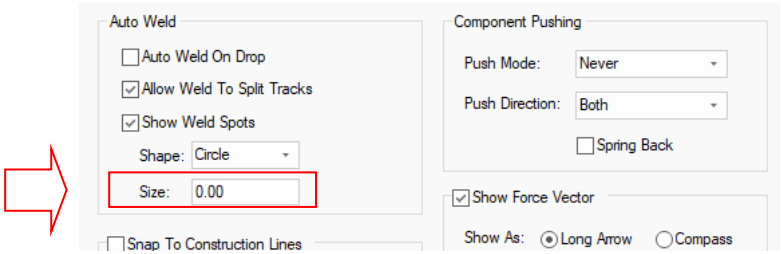


The example below shows a Square weld spot shape:



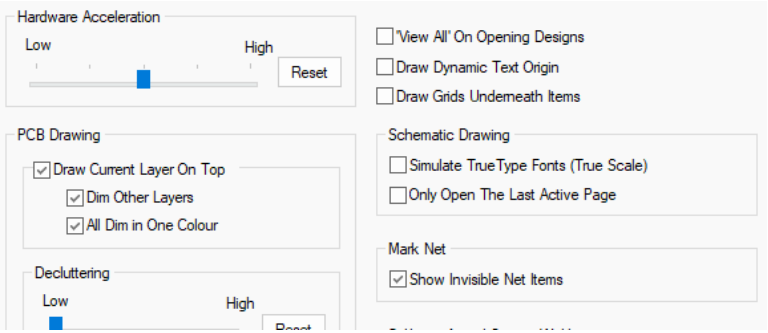
Weld Spot Size

In the **Options** dialog under **Move**, using the **Show Weld Spots** check box, you can now supply a size to draw the spots. If you set it to zero it will use the default size that was used previously. This was the default junction size in a Schematic design and 28 Thou (0.7112mm) in PCB.



Simplified Display dialog in Options

The **Options** dialog and **Display** page have been reorganised and simplified by removing redundant drawing options such as Fast Redraw and Use for Dynamic Items.



3D Viewer Changes

STEP 3D Viewer Performance Improvements

There are large performance improvements across generation and output of the STEP 3D Viewer due to more efficient utilisation of multi-threading resources. This results in improvements of an up to 80% faster generation time. On average, an improvement of 50% has been seen, with larger designs seeing much larger improvements.

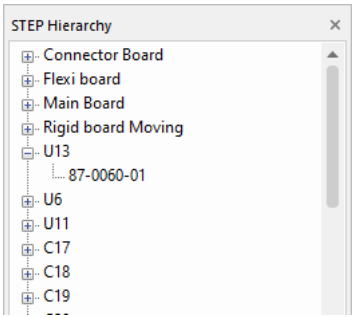
Calculation of flexi board folding has been significantly improved, also utilising multi-threading resources. Greater performance improvements can be seen with higher levels of fold animation.

STEP 3D Exploded View Performance Improvements

The performance of **Exploded View** generation in the STEP 3D viewer has been improved. Up to 90% increase in generation speed can be experienced with larger designs, with an average of 72% seen.

STEP Hierarchy Bar

Within the 3D Viewer, there is now a **STEP Hierarchy Bar**. This bar contains a tree that is populated with the internal document structure of a design in the STEP 3D Viewer.



This bar allows you to view all items currently in the scene, and enables the following functionality:

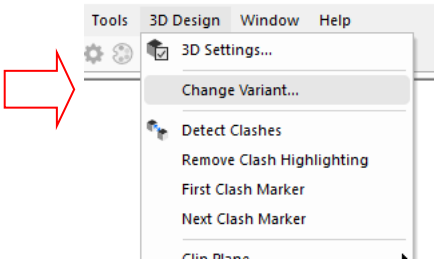
Selection – selecting an item in the tree will select the corresponding item in the 3d viewer, and vice versa.

Toggle Item Visibility – Context menu option with an item selected in the tree. This allows the user to selectively show/hide single items within the scene. Hidden items will still be output to STEP, as they still exist in the internal document structure, but will not be seen in the viewer itself. Hidden items will show in the tree with a dimmed text colour, so the user can differentiate between shown and hidden items.

View Extents – Context menu option with an item selected in the tree. This option will fit the view to show the bounds of the selected item in the 3D viewer, which allows users to easily find specific items.

Change Variant in 3D Viewer

Can now change variant in the **3D Viewer** using the **Change Variant** option from the **3D Design** menu.



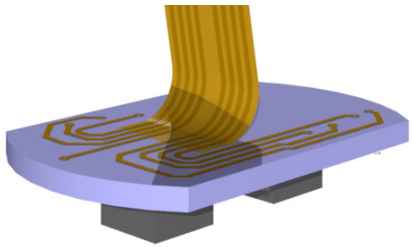
Changing the variant will update component shape, component footprint shapes, component areas, text and callouts within the 3D viewer. This removes the need to regenerate the entire scene to switch variants.

If the 3D design is open, changing variant within the PCB design will also update the 3D to reflect the changes.

Flexi Bend Region – Lift-off State

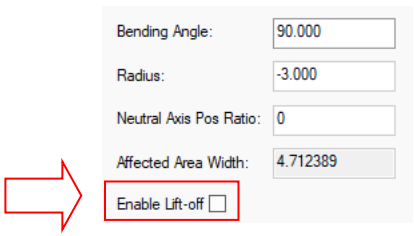
Flexi bend regions can now have a **Lift-off** state applied to them, this is set in **Bend Line Properties**. This will alter which transform is applied to a Rigid Moving Board that has bend lines within the board outline.

Lift-off means the flexi portion of the board can start within a existing board boundary and not exit through the 'edge' of the board.



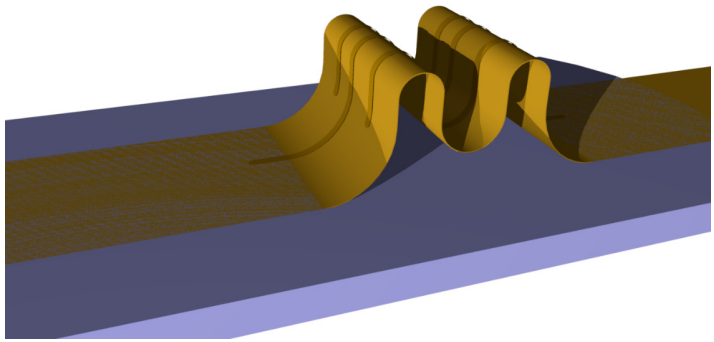
Flexi bend regions can now be placed on a flexi board within another board outline, if the flexi board lies on <Top Side> or <Bottom Side>.

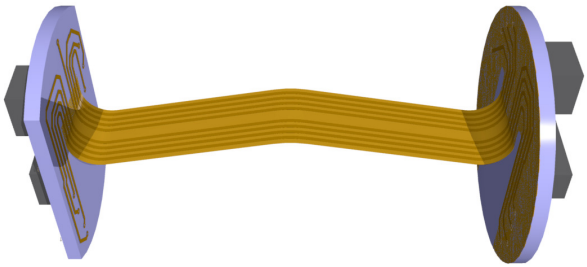
There is an **Enable Lift-off check** box for a flexi bend line, which can be set in **Bend Line Properties**.



Enable Lift-off – With this selected, the bend will lift-off the surface of the **Rigid Board**. The transform applied to a rigid board will be the same as the 'Bend' shape for this flexi board bend.

Below are some examples of what is now possible with this feature:

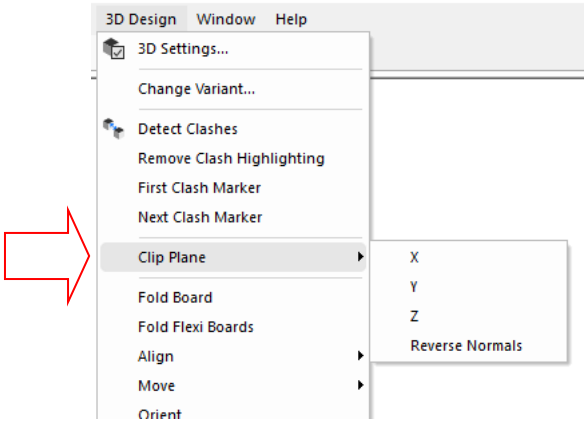




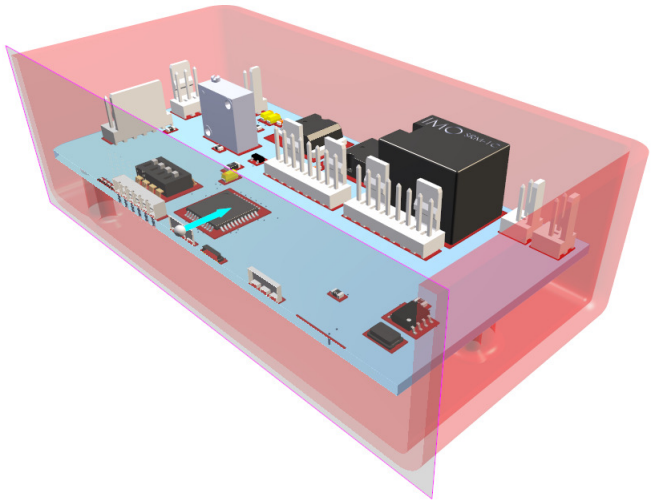
Interactive Clipping Plane

New **Clip Plane** options have been added to the **3D Design** menu for the **3D Viewer**.

A Clip Plane is a visualisation option that allows areas of the design to be selectively disabled from view enabling you to focus on areas of interest.



The clipping would look like this:



Options allow a clip plane to be added normal to the **X** axis, **Y** axis and/or **Z** axis, and allows interactive repositioning of the planes along their respective normal. You can use more than one plane view at any time.

The **Reverse Normals** option is used to reverse the X, Y or Z clip plane views by 180 degrees in the 3D Viewer.

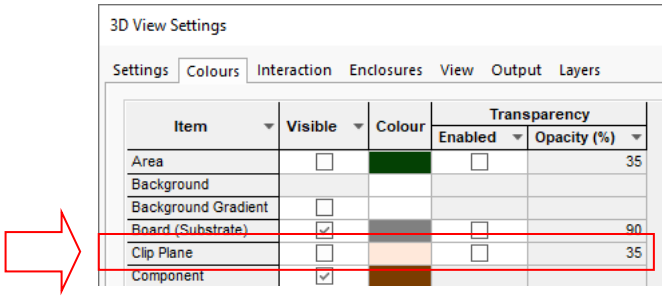
Once in this mode, you can drag the view using the interaction arrow.

To remove any Clip Planes from view, from the **3D Design** menu and **Clip Plane>** option, deselect the option to do not wish to view.

When using clip planes, they will automatically disable ray tracing in the 3D Viewer as this options is not supported. It will automatically reenable ray tracing if you remove all clip planes.

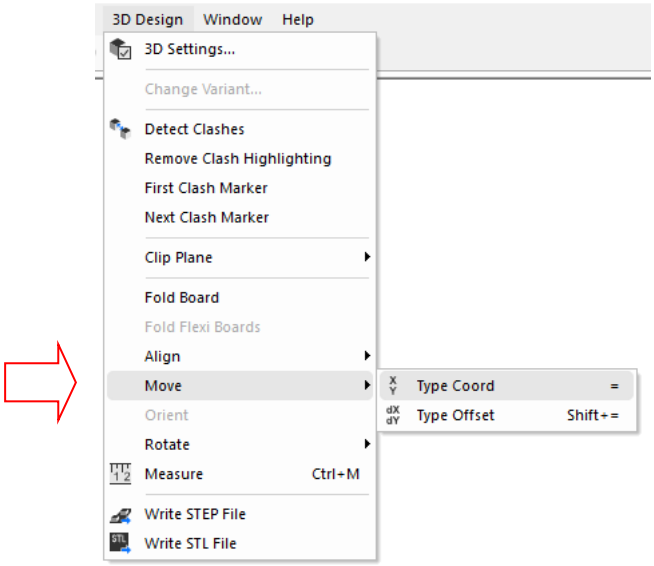
Clip Plane Colour

A Clip Plane colour has been added to the grid on the **3D Settings Colour** page, along with the option to enable transparency and the transparency value.



Type Coord and Type Offset in 3D Viewer

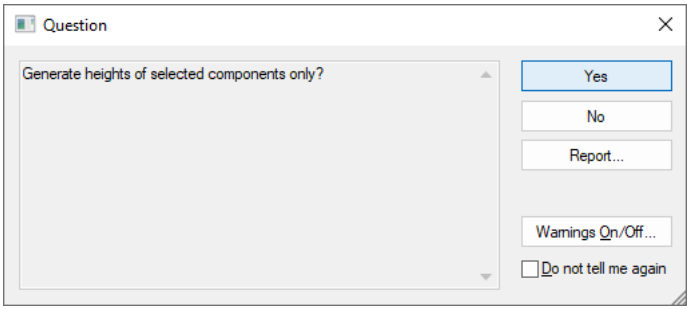
The **Type Coordinate** and **Type Offset** options can now be used in the **3D Viewer**. These options have been added to the **3D Design** menu under the **Move >** option.



An item in the viewer must be selected for the options to be available for use, these will be disabled if no item is selected. 3D objects can only be moved in the X,Y plane.

Generate Height from STEP for Selected Components

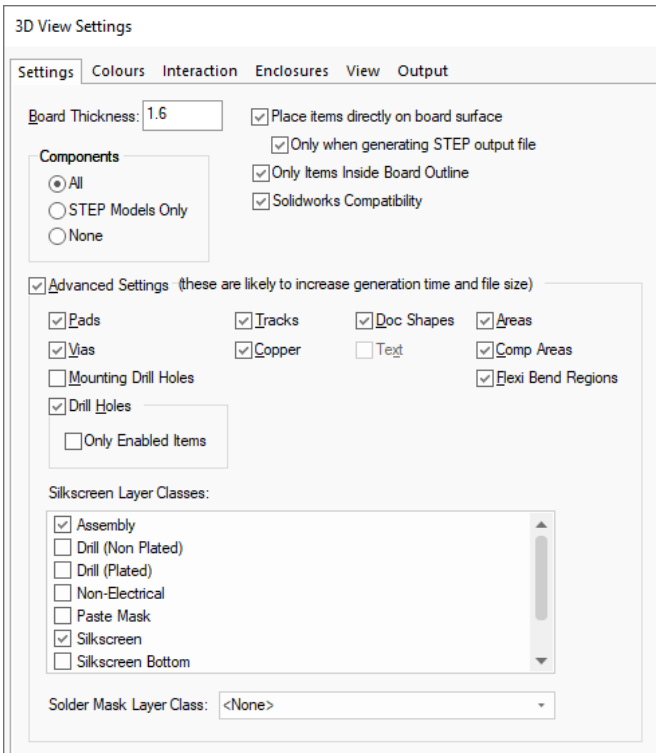
When using the **Generate Heights** from the **3D Design** menu with items selected, a warning message will appear asking if you would like to generate heights for only selected components.



If you click yes, only the selected components will have their heights generated from their STEP models. If you click no, Generate Heights from STEP will proceed as normal, generating all heights in the design.

3D Settings – Dialog Reorganisation

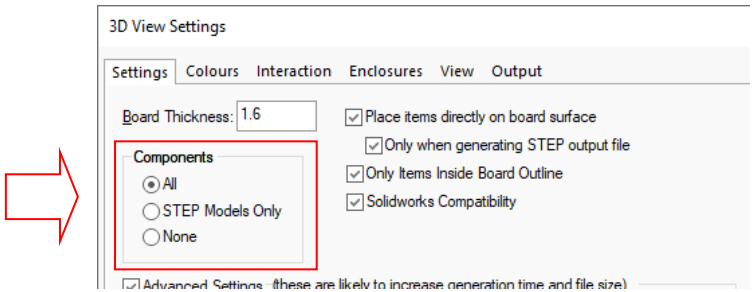
The **3D Settings**, **Settings** page has been reorganised to group the **Advanced Settings** into more logical groups. Other selective options have also been moved on the dialog.



Component Model Selection for 3D Viewer

The **No Components** check box on **3D Settings** dialog has been changed to a **Components** section.

This now has three radio buttons - **All**, **STEP Models Only** and **None**.



All will generate all the components as normal.

None will not generate any components.

STEP Models Only will only generate components with valid STEP models, and so will **not** use any 3D package definitions, or create a default package definition if one can't be found.

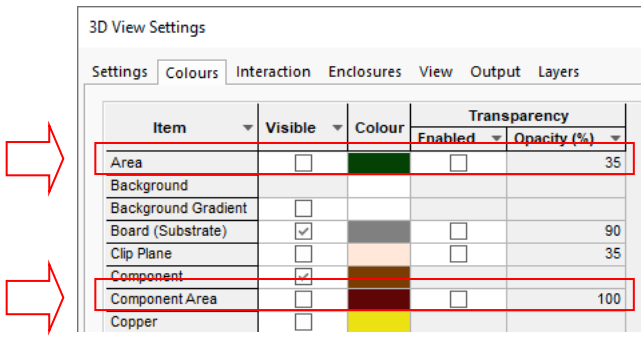
Note: Normal usage would be for each component to be exported using its associated STEP model, if one isn't available then it uses the 3D Package definition. If this isn't available, it uses the Component Height attribute. Finally, if this isn't available, it then uses a built-in system value of 2mm for the component height extents.

New Area Colours

The **Area** colour in the **Colours** grid in **3D Settings** has been split into two different colour selections for **Area** and **Component Area**. This allows you to differentiate between the two area types within the 3D viewer.

Area and Component Area colours can now have a transparency.

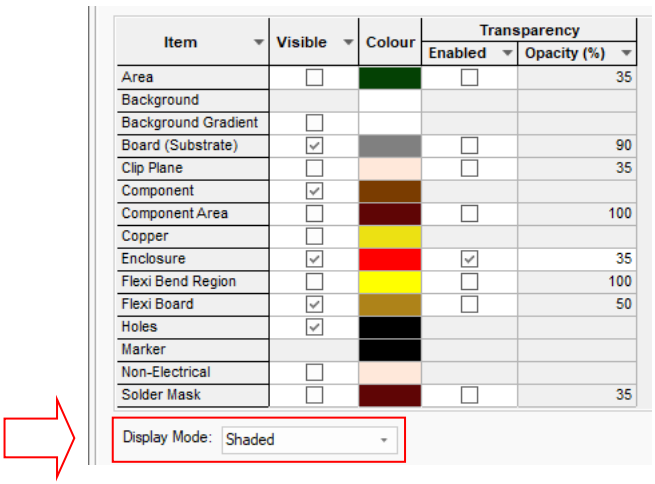
Note: to show areas in the same way as in previous versions, you can set transparency enabled with an opacity at 0 (meaning fully transparent) - this will show a wireframe-like version of the area.



To facilitate clash detection for areas, they can no longer be drawn as a wireframe. This is because clash detection requires triangulation, which is not calculated of wireframe shapes.

New Display Mode selection

The **Is Solid** check box on the **3D Settings Colours** page has been changed to a **Display Mode** list box with three modes available.

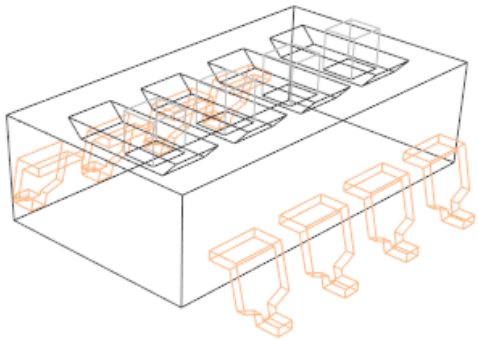


The previous, **Is Solid,** option allowed changing the display mode of the viewer between Shaded and Wireframe modes.

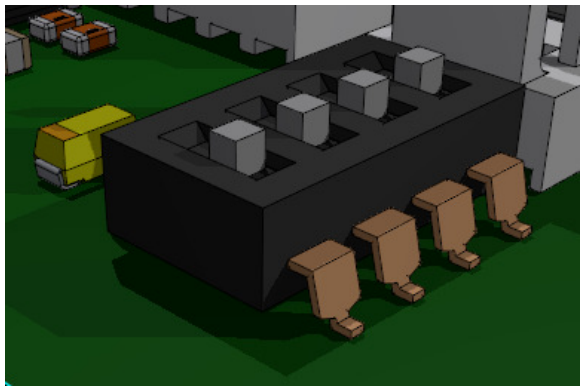
The **Display Mode** list contains both **Wireframe** and **Shaded** modes, but also **Shaded With Edges** mode. This mode is essentially Wireframe and Shaded modes combined – displaying the shapes as shaded, but with face boundary drawing enabled.

When changing display modes, it will no longer require a full regeneration of the 3D Viewer, and will change the display mode for all shapes in the scene on-the-fly.

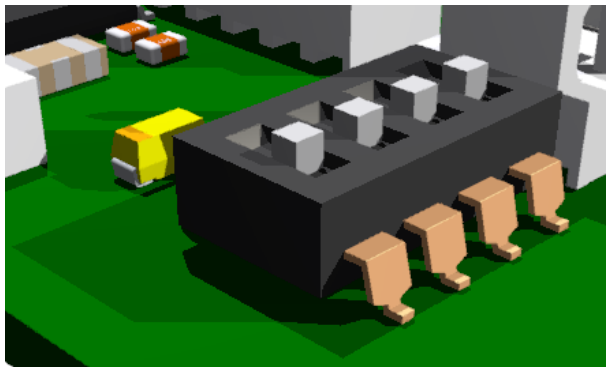
Wireframe only displays like this:



Shaded with Edges displays like this:

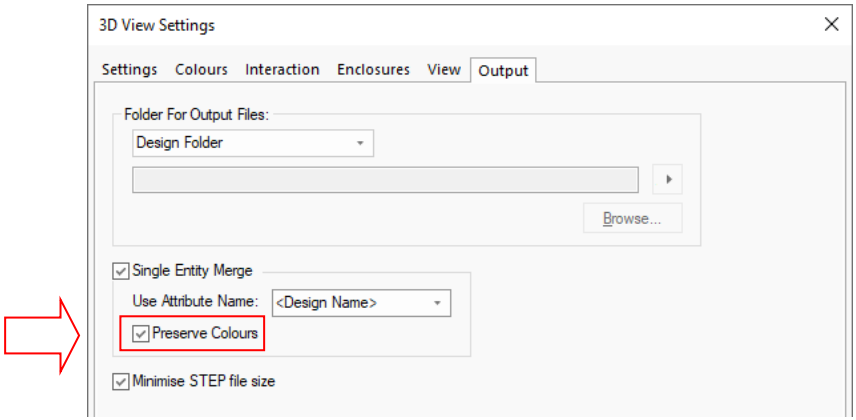


Shaded displays like this:



Single Entity Merge – Preserve Colours

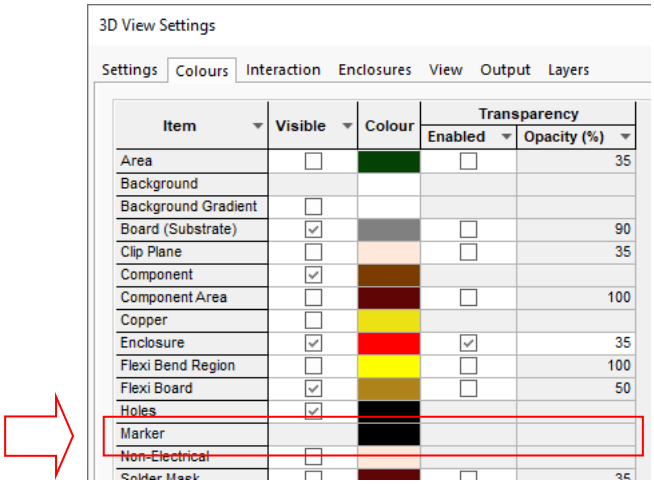
A **Preserve Colours** option has been added to the **Single Entity Merge** section on the **3D Settings Output** page. Enabling this option will retain the colours of the shapes in the scene when merging them into a single entity.



Note: This has been added as a separate option because the merge process needs to be less destructive to be able to map colours correctly. With Preserve Colours enabled, the file will be slightly more complex (includes colour definitions and more complex geometry definitions). With Preserve Colours disabled, the geometry can be simplified further, resulting in a slightly smaller file size.

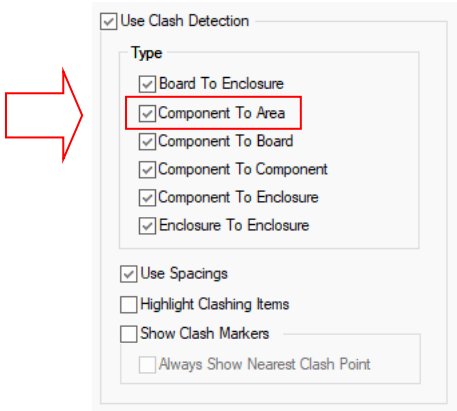
New Measure and Clash Marker Colours

The **3D Settings Colours** page has been changed to add a colour selection for the **Measure** and **Clash Marker** options. These colours will also be used within the **3D Viewer** and **Position STEP Model** options.



Component Keep Out Clash Detection

A new **Component To Area** option has been added to the **Clash Detection** section of the **3D Settings, Interaction** page. This will enable clash detection between components and component keep out areas.



For an area, if no height is defined, components models clashing with the area at all will be classed as a clash. If a keepout height is defined (**If Higher Than** selection in **Area Properties**), a clash will be found if the component is clashing with the area but only if the component is taller than the defined area.

Changes to 3DIgnoreClash STEP Attribute

The **3DIgnoreClash** user attribute has been replaced with an in-built Part attribute **<STEP Ignore Clash>**. Using a built-in attribute means you can add it to a **Component** using **Properties** and select it from the drop down list.

Any existing designs that used the old method are converted to use the new attribute when opened and subsequently saved also when the design is saved.

New <STEP Suppress> Attribute

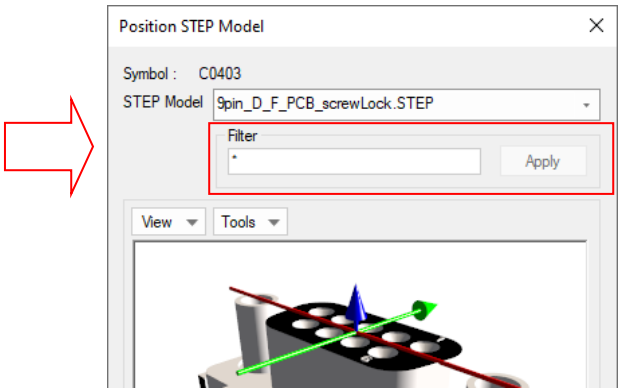
A new <STEP Suppress> attribute has been added, this can be added to a component. A component with this attribute will not be generated in the 3D Viewer or output.

This allows you to selectively suppress components that they do not require, without needing to use an empty <STEP Filename> attribute.

If you use <STEP Filename> attribute (with a filename) and also add the <STEP Suppress> attribute, this overrides the <STEP Filename> attribute and suppresses it in the 3D Viewer and the output.

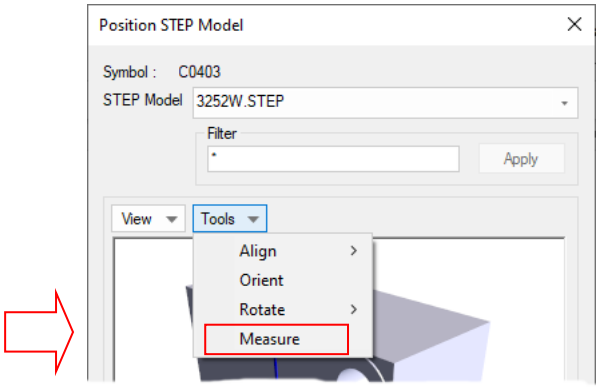
STEP Model filter in Position STEP Model dialog

There is now a **Filter** section in the **Position STEP Model** dialog that allows you to apply a filter to the STEP Model list to refine it.



Measure option in Position STEP Model

The **Measure** option is now available in the **Position STEP Model** dialog (for Footprints and Enclosures) on the **Tools** drop down menu.



This works exactly the same as **Measure** in the **3D Viewer**. If you wish to select a specific face, use the Shift key while making your selection.

The colour of the measure marker can be changed to your own colour, see below.

Outputs using Direct2D Drawing

The **Print**, **Capture Window** and **Embedded View PDF Output** functionality in Pulsonix can now use Direct2D drawing technology and so can appear exactly the same visually as when displayed on screen, showing such drawing effects as translucency and patterning.

This is applicable when using **Hardware Acceleration** levels 3 - 5 (in **Options**, **Display**).

New command to Rotate Clockwise One Step

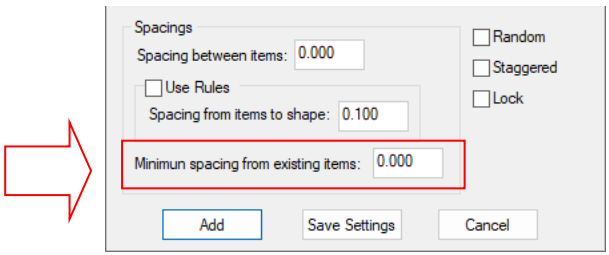
A new command has been added to **Rotate one step clockwise**. There is no interface change for this feature, but it can be assigned to a shortcut key - use the **Clockwise Rotation** option.

Apply Vias/Pads – New Options

New options have been added to the **Apply Vias/Pads** option to extend its functionality.

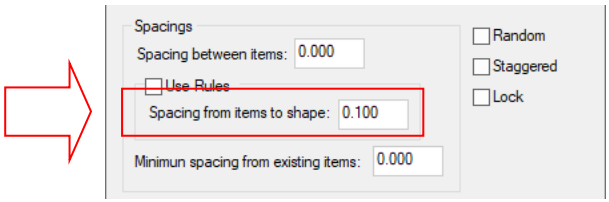
Minimum spacing from existing items inside the shape

For **Fill The Shape** option, you can now set the **Minimum spacing from existing items**.



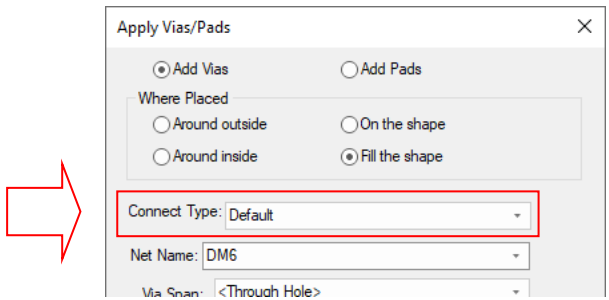
Spacing from items to shape for Filling the Shape

Spacing from items to shape is now available for **Fill The Shape** option.



Connect type

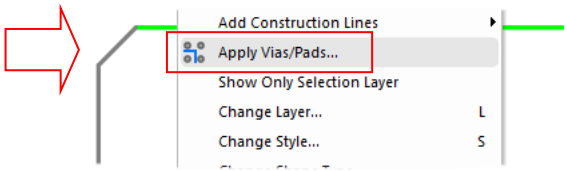
For the Vias setting, there is now a new Connect Type box that allows you to define the connect type of the Vias. Selections are available for **Default**, **Isolated**, **Not Isolated** and **Thermal Pad**.



Staggered Option in Apply Vias/Pads

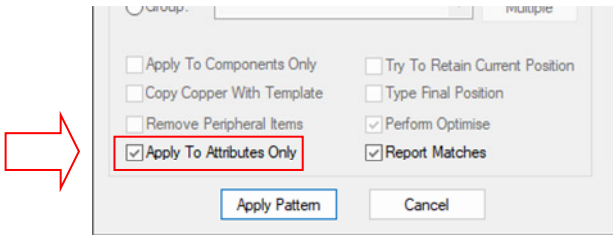
The **Staggered** option is now available for open shapes when **Around Outside** option is selected.

The **Apply Vias/Pads** option available on the context menu when copper or a track is selected



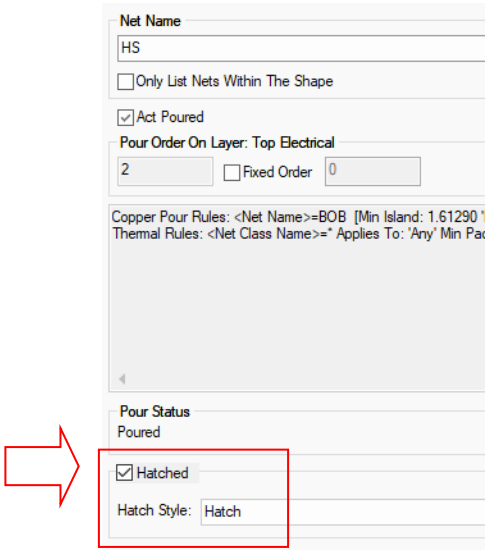
Apply Layout Pattern Apply to Attributes Only

The new **Apply to Attributes Only** switch on the **Apply Layout Pattern** dialog updates the position and rotation of common attributes only between matching Components.



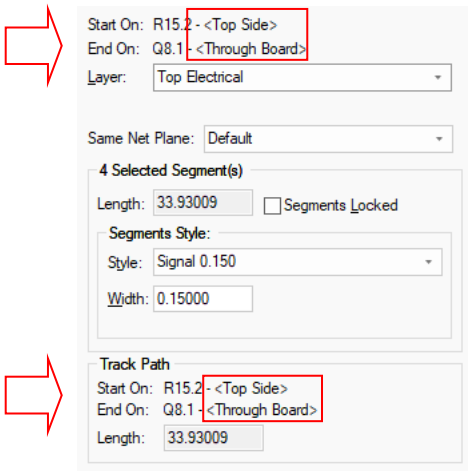
Hatch Style Override in Template Properties

On the **Template Properties** dialog and **Template** page, a new **Hatch** button and **Hatch Style** box have been added. Selecting this will override the **Copper Pour** rules and use the selected hatch style from the list box instead.



Track Properties – Display the Layers of Start and End Nodes

In the **Track Properties** dialog, in the sections for **Start Node** and **End Node**, the layer of each node is now displayed, (separated by a -). For example Start In: R15.2 - <Top Side>

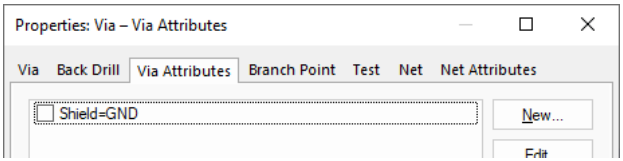


Show Only Selection Layer

A new command has been added to the context menu to **Show Only Selection Layer** for layer based items, such as tracks. It is only available if the item you have selected has a layer and the layer isn't a span or a layer set.

Via Properties – Ability to add Via Attributes

You can now use the **Via Properties** dialog to add **Via Attributes** to a via. You may use this to create specific rules for Vias, Spacing rules for example on vias that are used for shielding.



New Cutout Text option

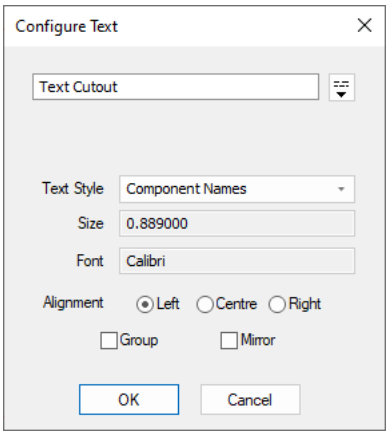
You can now add stroke, true-type and special text as shape **cutouts** and merged shapes.

For example, you can add layer designation to a copper layer:

To use this option

Select **Cutout** from the **Insert** menu. Right click and select **Insert Cutout Text**.

Type the text required in the dialog:



Select the shape you wish to add the text cutout to, select the whole shape.

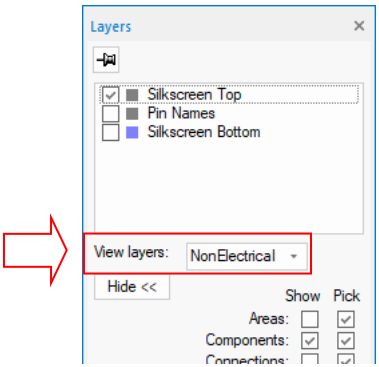
*Note: You could also start with an unselected shape and select the **Insert Cutout Text** option first, then select the shape to apply it to.*

Position the text and release.

It should be noted that the **Copper Line Style** should be a smaller line size so that the text shape can be properly cutout as intended.

Layers Bar – View Layers option

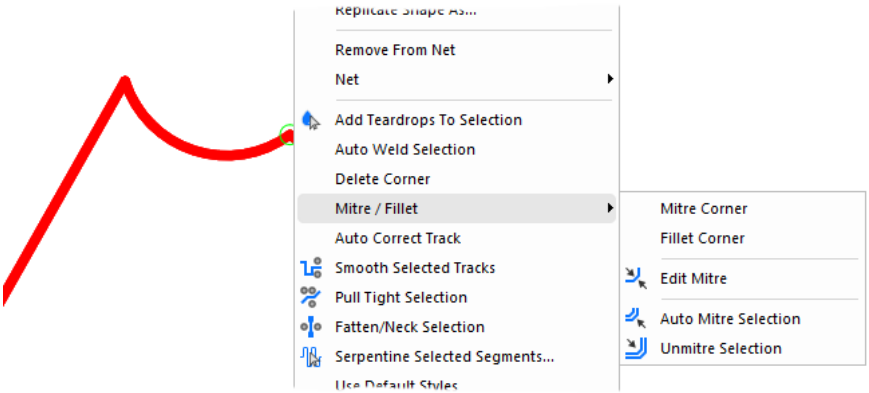
In the **Layers** bar, there is a new option to **View Layers**. This enables you to filter the layer types to display in the Layers list. The list box allows you to select between **All**, **Electrical** and **Non-electrical**.



Fillet/Mitre on Corners with an Attached Arc

You can now use **Mitre Corner** or **Fillet Corner** when they are attached to **Arcs**. These options are available from the context menu for a selected corner.

Right click on a corner:



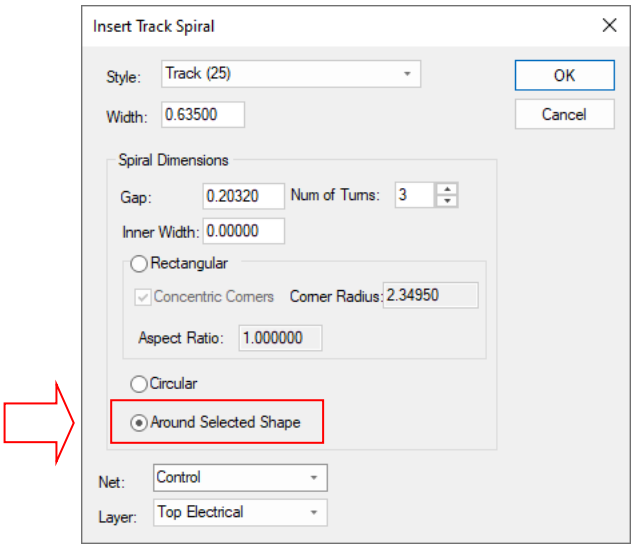
The resultant corner now looks like this:

When using mitre on arc ends, it may result in a mitre of any angle.

This feature is also available when working interactively within the **Edit Mitre** option.

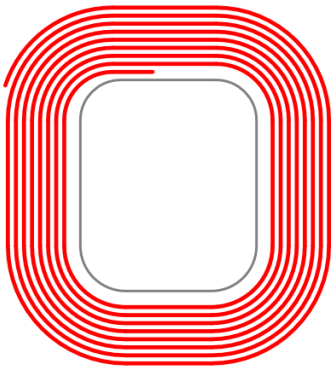
Insert Spiral – Spiral Around Shape

From with the **Insert Spiral** dialog, you can select the **Around Selected Shape** option. This new mode enables you to dynamically spiral around shapes for a PCB design or footprint. This can be performed for tracks and copper shapes.



With this option selected, After pressing OK, the dialog will disappear from view and you will be able to hover over items in your design.

If a suitable shape is available, the cursor will change to show the spiral symbol, otherwise, the cursor will turn to an X. A shape is typically invalid for a spiral if it's not closed.



You can change the shape to be **Closed** using **Properties** of the selected shape.

You can also spiral around pre-selected shapes. Select a shape you want to spiral around, and enter the Spiral (Tracks or Copper) dialog from the **Insert** menu.

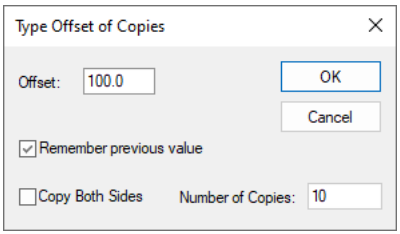
Dynamic Snapping when Editing Shapes

When **inserting** and **editing shapes**, it now visually shows you the item that it is snapping to as you hover over other shape items. Previously would only snap after placing the shape.

The **Snapping to Item** option is available on the context menu during editing.

Construction Lines - Copy With Offset Dialog Change

When using **Construction Lines**, in the **Copy With Offset** dialog, the label **Copies** has been replaced by **Offset**, and **Steps** had been replaced by **Number of Copies**. These changes make the dialog more logical for use.

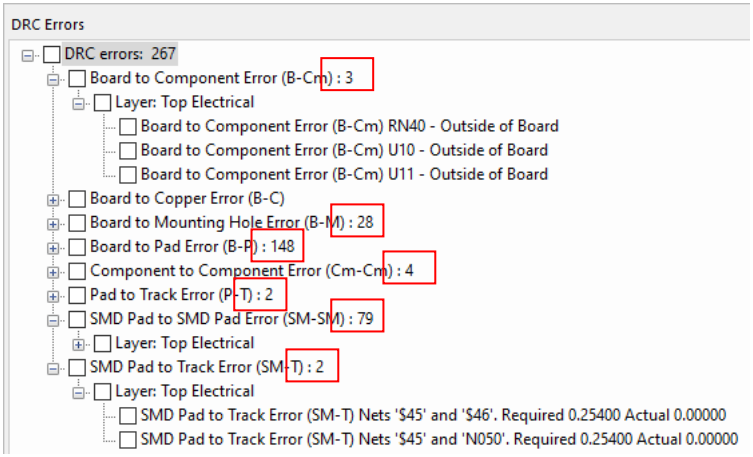


This feature was back-fitted to V13.0.

DRC Changes

DRC Error Bar – Sub-totals of Errors Found

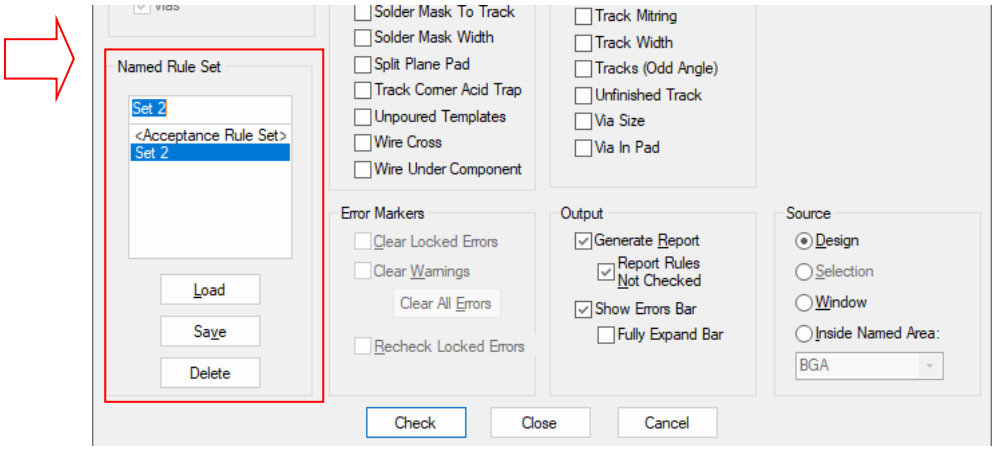
For each found error category, the error count is now displayed next to the error description.



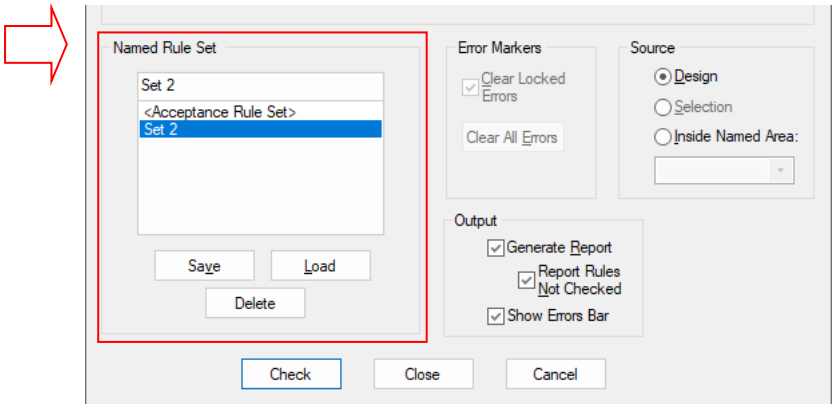
DRC Named Rules Sets (Acceptance)

In the DRC/ERC dialogs, there is a new **Named Rule Set** box list that replaces the Acceptance Rule Set. From here, you can now define different rule set names.

The PCB DRC dialog:



The Schematic ERC dialog:



By default, the dialog has the <Acceptance Rule Set> that always appears in the list. This can be used or you can also define your own sets for different checking purposes. For example, a set that is used for Spacing checking during design, and then a full check at the time of plotting for a full verification of the design.

Once the check box selections you require have been made, to add a new rule set, simply type a new name in the entry box and press **Save**. This and the default set will be saved into the design.

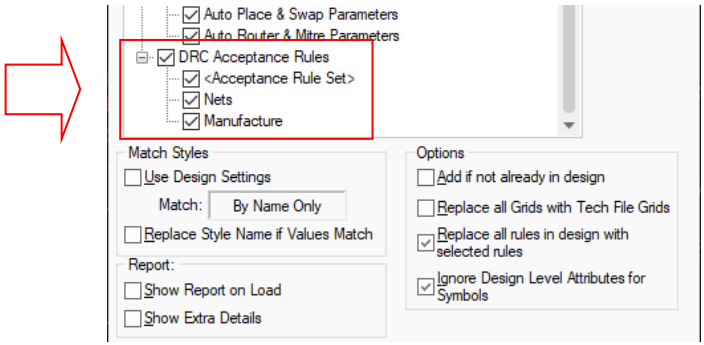
If you make switch changes and press **Save**, the currently selected named set will be updated.

Likewise, a selected named set can be deleted by pressing the **Delete** button.

By changing the string on the combo box and pressing save, a new rule set name is created with the current rule checks. The rule set names are saved in the design.

Load/Save Technology

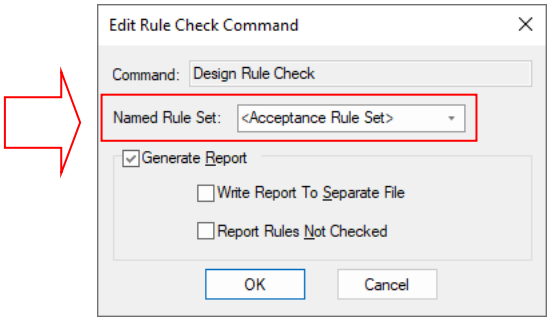
The **Load/Save Technology** options now lists all the available rule set names that are saved in the Technology file.



Report Maker – ERC/DRC command – Named Sets

The ability to use a Named Rule Set as an acceptance set within the **Report Maker**, **Electrical Rules Check** and **Design Rules Check** commands has been added.

A named rule set list box has been added to both commands. This will list any existing named rules found in the design.



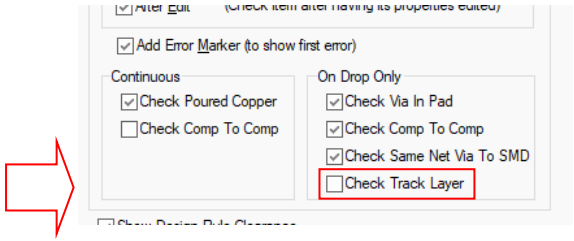
The default named rule set, <acceptance rule set> is used if you do no specify your own rule set.

ERC/DRC in CAM Plots dialog

When using Report Maker format files within the CAM Plots dialog, you can use multiple named rules sets when using the **Report Maker**, **Electrical Rules Check** and **Design Rules Check** commands.

Online DRC – Check Track Layer Option

On the **Online DRC** page of **Options**, there is now a **Track Layer check**.



When enabled and tracks are not allowed on certain layers (using the **Track and Via Size Limit Rules**), when you change to one of these layers and try and finish the track, it will report a track layer error (T-L).

If you have **Continuous Online DRC** enabled, you will get the Track Layer error as soon as you pick the layer and exit out of the **Change Layer** dialog.

This feature was back-fitted to V13.0.

Improved Precision of Silkscreen Overlap DRC Check

Silkscreen overlap DRC check previously only checked a bounding box around the text, leading to false positive DRC errors, where the text was not actually overlapping the pad. The check will now determine if individual text shapes overlap the pad, and so will only produce an error if the text truly overlaps the pad.

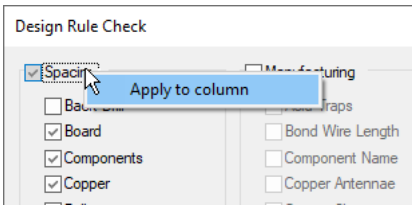
The example below shows how a pad sites inside the text bounding box without producing DRC errors:

The red area is the invisible text bounding box. The pad clearly sits inside without creating an error.

This feature was back-fitted to V13.0.

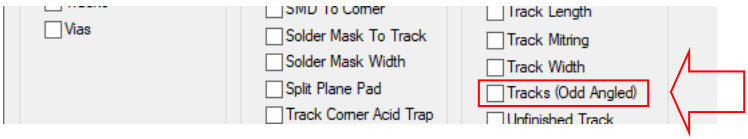
Apply to Column

Within the DRC dialog, you can right click on any header of a column and use the **Apply to column** from the context menu. This enables you to apply the current header setting to all options within that column.

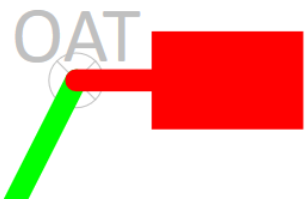


Odd Angled Track Check

A new **DRC** check has been added to detect odd angled tracks (any angle that is not 45 or 90 degrees). This is available under **Nets** and **Tracks (Odd Angled).**

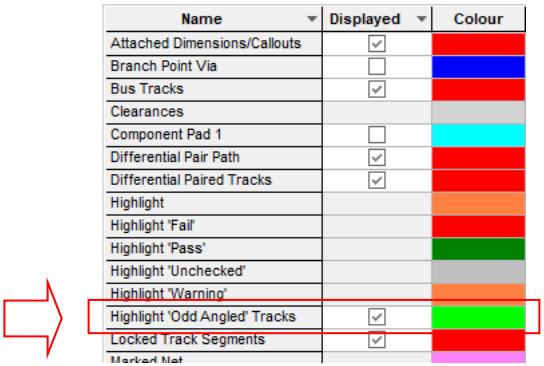


When run, if an error is detected, an OAT error marker is added to the design. The error is displayed on the start point of any track that has an odd angle. This check is in addition to the **Odd Angled Track Colour** highlighting (see above).

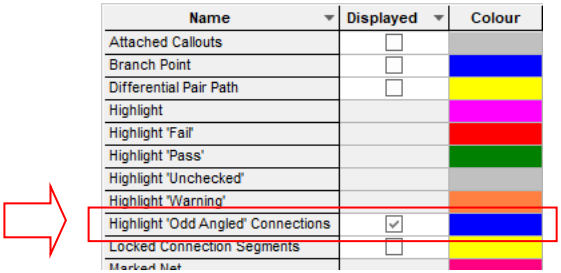


Colours - Highlight Odd Angled Tracks / Connections

In **Colours**, **Highlights**, there is a new colour to **Highlight 'Odd Angled' Tracks**. If the check box under **Displayed** is checked, then any offending tracks will be highlighted if the angle is not 45 or 90 degrees.

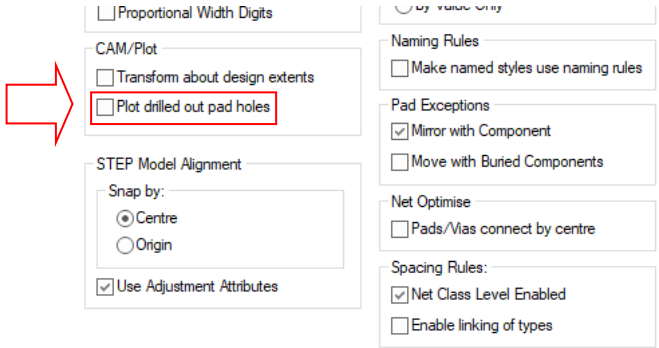


In Schematics, the option is **Highlight 'Odd Angled' Connections** and will highlight connections if the angle is not 45 or 90 degrees.



Design Settings – Save default 'Plot Drilled Out Pad Holes' status

You can now set the default status of **Plot Drilled Out Pad Holes** for use in the CAM Plots option. This setting will be saved with the design. The default setting is saved in the **Technology** under **Design Settings**, **General** and **CAM/Plot**.

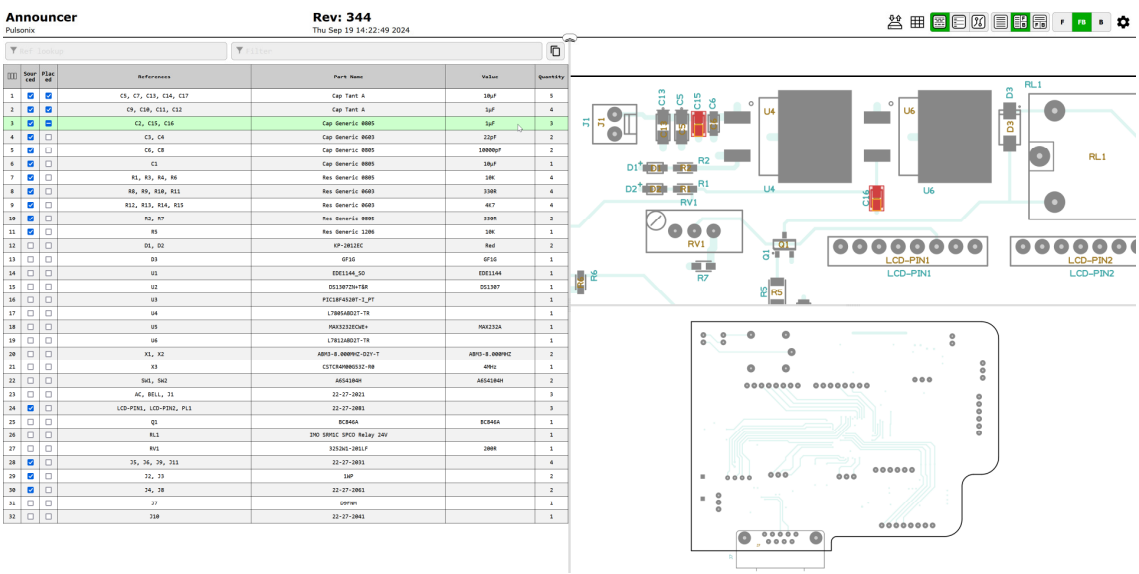


This feature was back-fitted to V13.0.

Interactive HTML BOM Option

Support has been added for the **Interactive HTML BOM** option. This tool is a plugin option that exports limited Pulsonix design information into a HTML format that produces an interactive design and BOM listing that can be viewed and searchable in a web browser. Only the Top and Bottom layers are exported and BOM data. No inner layers or attribute details are exported.

This tool is particularly useful if you are producing boards in-house and hand soldering prototypes. It will guide to finding Parts with the same value and their locations. Items selected in the BOM list are highlighted in the design view.



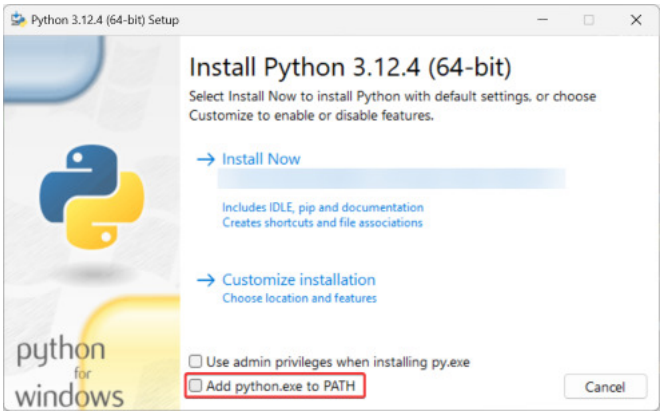
Pulsonix automatically generates an input file that is read into the browser ready for use. You must have Python installed and configured first though for this to be usable.

Python 3.8

The generation of the Interactive BOM requires **Python 3.8** (or later) to be installed.

You can download the Python installer here https://www.python.org/downloads/

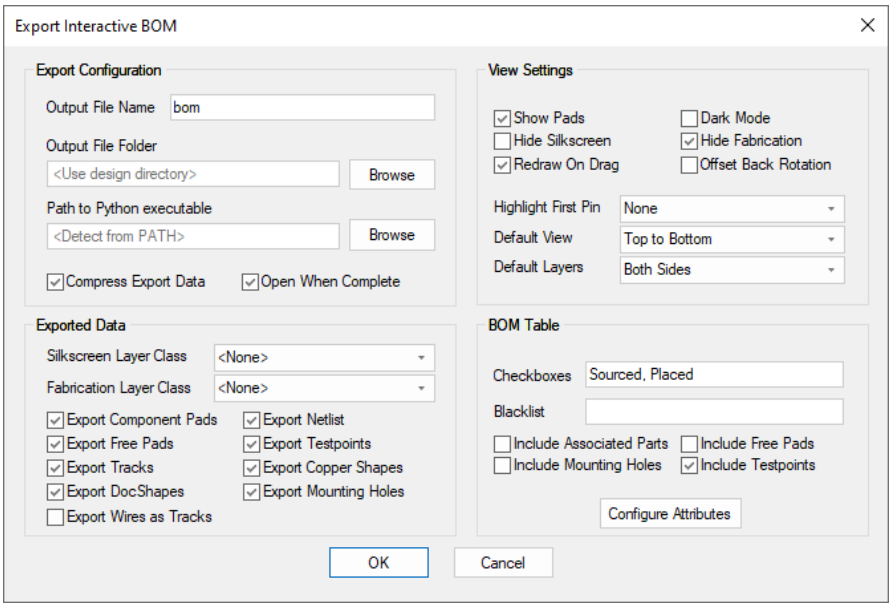
If you installing Python 3 for the first time, it is recommended that you select the **Add python.exe to PATH** option in the installer, this will allow Pulsonix to automatically locate your installation.



Once successfully installed, the option is ready to run.

Using the Interactive BOM option

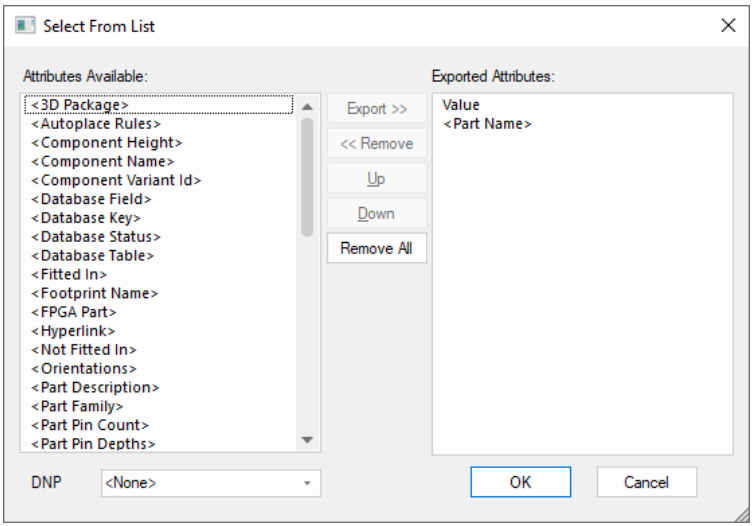
From the **Output** menu, select the **Interactive BOM** option. You are presented with a dialog.



Once configured, pressing **OK** on the dialog creates a HTML file that is then automatically sent to your web browser for viewing. You must have the **Open When Complete** switch selected on the dialog for this to happen. The HTML file saved is self-contained and can be used at a later time or given to another person on another machine (provided they have Python 3.8 or later installed).

Configure Attributes

Selecting the **Configure Attributes** dialog enables you to select the attributes required:



Import Allegro PCB

If you have the Cadence Allegro Import option enabled in your licence, you can now import files with the .alg file extension that have been generated by the Allegro Extracta program.

The Extracta.exe program is supplied with Allegro but is run outside of their PCB environment but still as part of their supplied program suite. Pulsonix calls this program in order to export an ASCII file that can then be read in. In this instance, both Allegro and Pulsonix must reside on the same machine.

This new alternative mechanism enables files generated on **another** machine using the Allegro Extracta.exe program to be loaded into Pulsonix. In order to extract the data required there are a number of actions required:

Locate the batch file and associated .txt file supplied with your installed Pulsonix (usually C:\Program Files\Pulsonix 14.0\SysUtils\

Copy **AllegroToPulsonix.bat** and the **AllegroTranslate.txt** control file to the machine that has Allegro installed. Copy to an accessible folder (i.e. not the Programs folder containing Allegro). For example, create a folder called C:\Temp\

Copy the Allegro PCB design file (.brd) that you wish to convert, into the same folder as the batch file.

Launch a Windows command shell (type CMD in the Windows Start menu and press <Enter>).

In the command shell, navigate to the folder that contains the batch file (i.e. cd C:\Temp\ )

Run the AllegroToPulsonix.bat using the following command in the command window:

AllegroToPulsonix design_name.brd <Enter>

Note, normal command shell rules apply, for example, if you have spacing in your filename, you will need to enclose these in quotes "design name.brd"

Running the batch file will create an ASCII file of your design with the .alg extension (i.e. design_name.alg).

Copy the resultant .alg file back to where Pulsonix can read it in.

In Pulsonix, select the **Open** dialog on the **File** menu, there is a new file type. **Allegro PCB ASCII File (*.alg)**. Select your .alg design file for import.

The Pulsonix import filter will now run the Allegro import option.

External Resources

If you require an Allegro design in ASCII format, you can provide the AllegroToPulsonix.bat and the AllegroTranslate.txt control file to the person performing the extraction. The batch files does not require Pulsonix in order to be run. Once run, the resultant .alg file will be provided back to you for import.

Graphical Design and Symbol Comparison

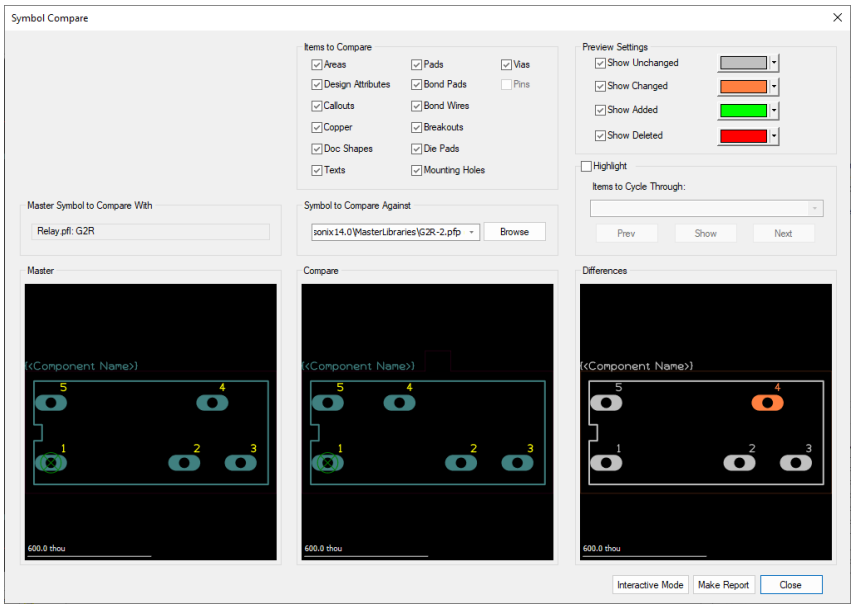
There are two new modes for design and symbol comparisons:

The **Symbol/Footprint Compare** option will compare two symbols together and report or show you the differences. Differences can be shown within the dialog or 'overlaid' within a new interactive comparison mode. The **Compare Against** option is available from the **Tools** menu

The existing **Design Revision Analyser** has been updated so that the **Interactive Mode** can be access from within the dialog. This option is still available from the **Tools** menu.

Symbol Compare

Run **Compare Against** from the **Tools** menu. Works for both Schematic Symbols and PCB Footprints. The original symbol would need to be saved as a symbol file for a comparison.



Three preview windows are displayed:

Master – the left-most preview is the symbol currently being edited.

Compare – the middle preview is the symbol file being compared to; the one defined in the **Other Symbol to Compare Against** entry above it.

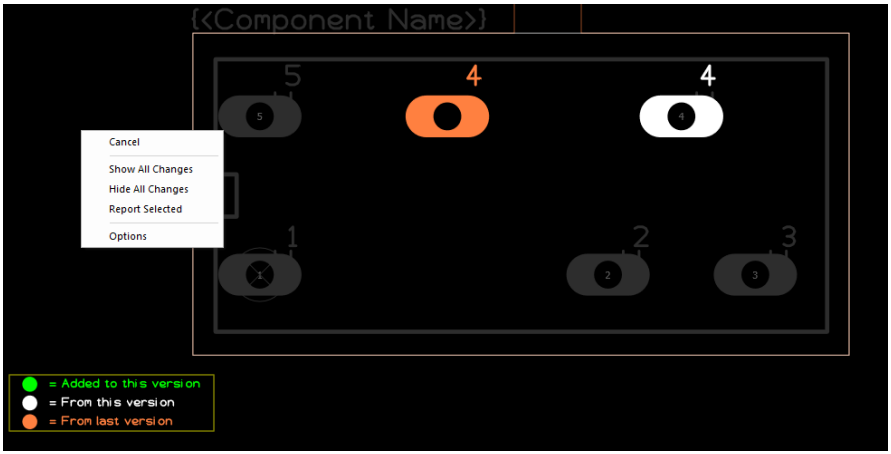
Differences – the right-most preview shows you the differences found of the two symbols.

Other settings on this dialog enable various facets of the symbol to be **compared** and displayed and for the **Preview Settings** colours to be defined.

The **Highlight** check box enables **Items to Cycle Through** to be selected. You can choose from **Areas**, **Design Attributes, Doc Shapes**, **None**, or **Pads**. Once an item is selected from the list, the **Prev**, **Show** and **Next** buttons will be enabled for use in the preview.

Interactive Mode

Once in the interactive mode, both the Master and Compared symbols are displayed.



Mode legend

A legend is displayed within the symbol design area. This shows you the colour coding for the different levels of comparison.

Added to this version

From this version

From last version

When using the interactive mode, at any time you can right click and from the context menu, options are available for use:

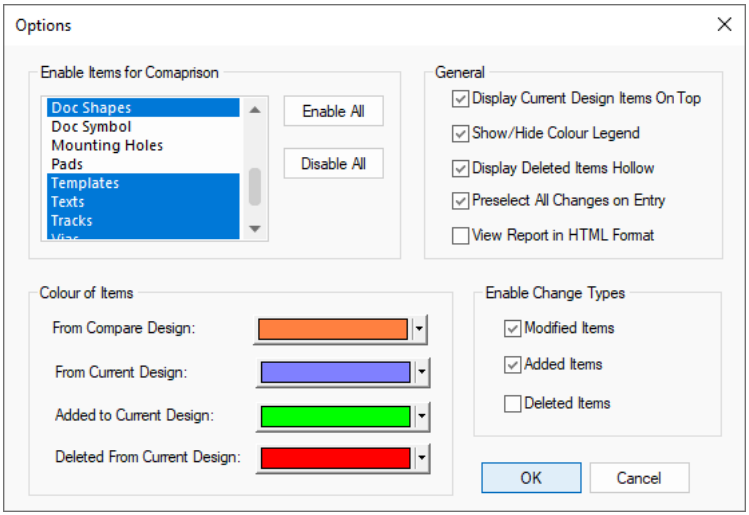
Cancel – selecting this will exit the mode and put you back into the **Compare Against** dialog.

Show All Changes – This will display everything that has changed.

Hide All Changes – This will hide everything that has changed.

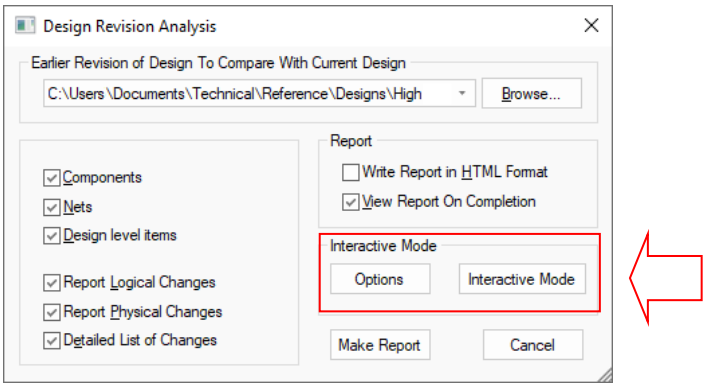
Report Selected – Hovering over an item in the design shown as a difference will display a small modal cursor. It enables you to select the item that has a difference. From here, a report of that item can be made.

Options – This displays options that are available when in this mode:

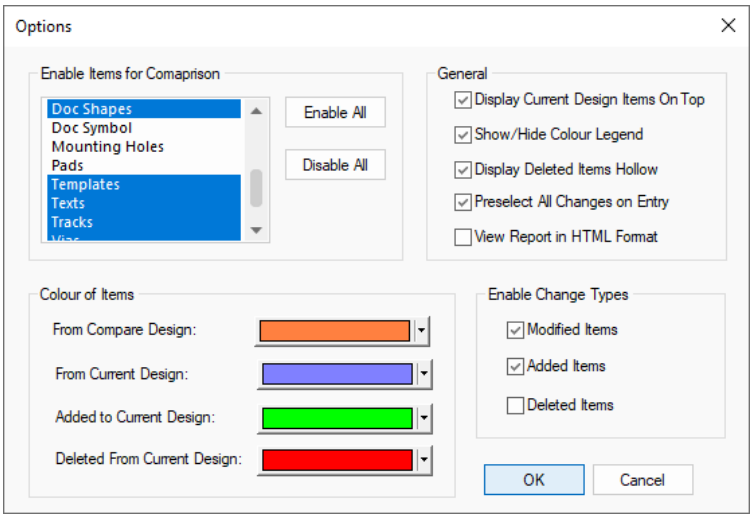


Design Revision Analyser

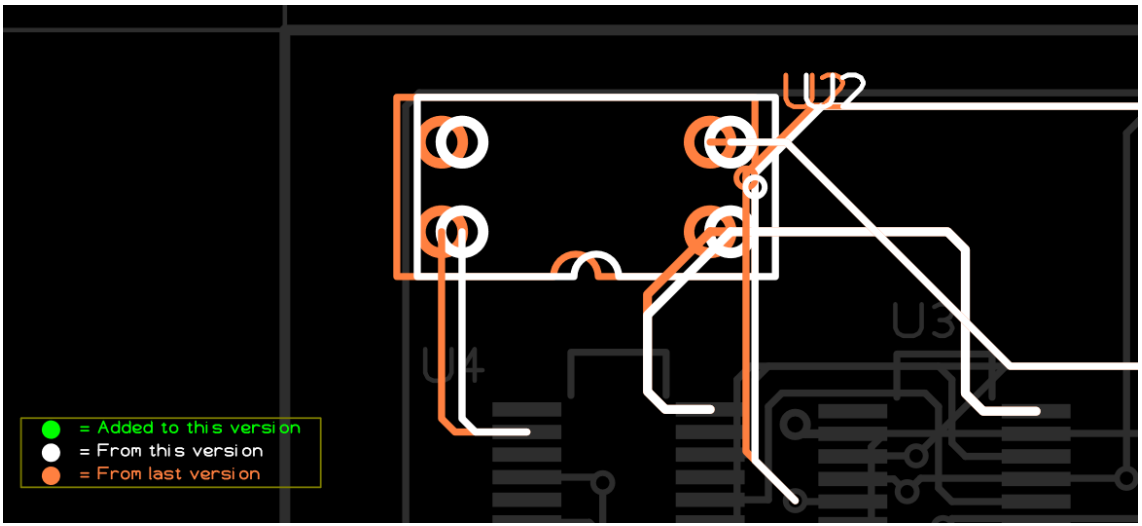
Changes have been made to extend the **Design Revision Analyser** so that as well as a report, a graphical comparison can be displayed using the **Interactive Mode** button from the dialog.



The **Options** button enables access to a dialog that allows you to define items that will be used in **Interactive Mode**.



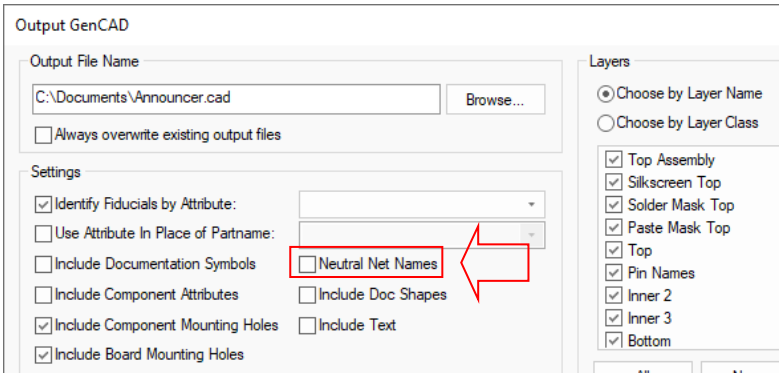
As with compare Against, the **Design Revision Analyser** in **Interactive** mode displays the differences:



GenCAD Export

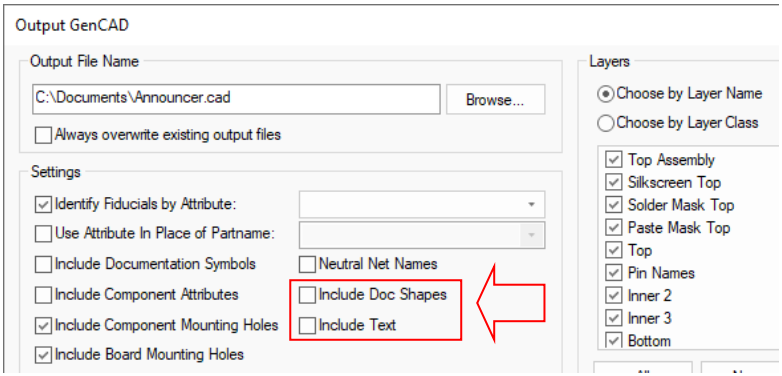
Neutral Net Names option

From the **Output** menu, the **GenCAD** option has s a new option for **Neutral Net Names**. When this is checked, all user defined net names will output as numerical net names. This helps you protect your design IP if sending the file to an external source.



New options to Include Doc Shapes and Text in Export

From the **Output** menu, the **GenCAD** option, there are two new check boxes to **Include Doc Shapes** and **Include Text**. If selected, doc shapes and text in the design, in components and in doc symbols will be output.



Vault Changes

Using the Vault in V14.0

In order to access the new Vault features below in Version 14.0, the following changes are required:

Vault Database version

For Pulsonix version 14.0, you should **update** your Vault database to version **1008** using the **Vault Setup** dialog. This dialog will also confirm which version you are currently running.

ODBC Driver version

The **V16.0 64-bit ODBC driver** is the latest version and will be automatically installed during the main Pulsonix product installation. It is not updated if it is already using this version as was installed with Pulsonix Version 13. This version of the driver is required on the Pulsonix client side to support the Postgres server V14.5.

Postgres version

For Pulsonix version 14.0, the PostgresSQL version has not changed and is still **14.5**.

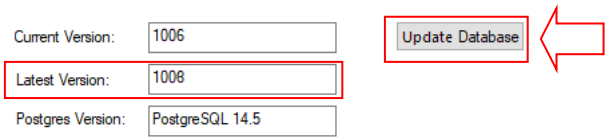
Updated Vault Database (to version 1008)

For Pulsonix version 14.0, all new or modified functionality listed below requires a new Vault Database to be installed. You should update your Vault to version **1008** using the **Vault Setup** dialog. This dialog will also confirm which version you are currently running.

Before updating, it is highly recommended that you make a backup using the **Vault Admin** program.

To update the database, from the **Setup** menu, select **Vault** (setup) and **Version** tab. You must be signed in as Admin to update the database.

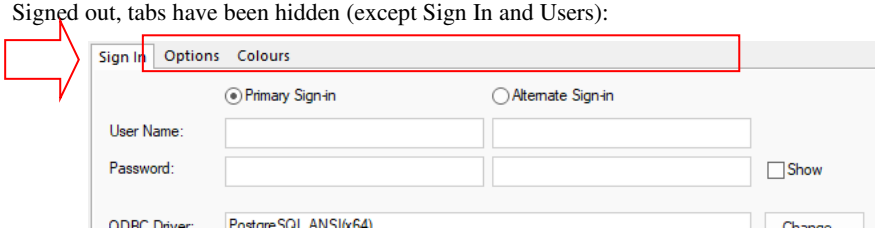
The **Update Database** button will be available for selection. The **Latest Version** button will display **1008**, showing you that the update is available.



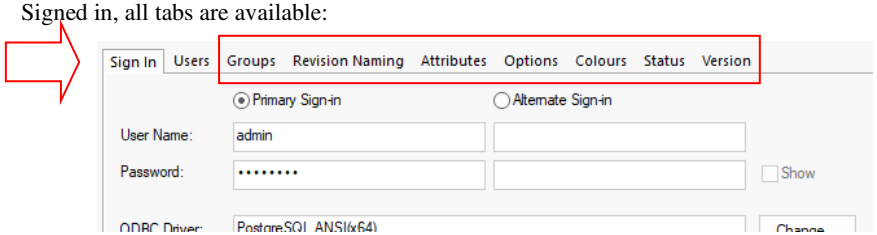
Vault Setup – Visibility of tabs reflects the Sign-in status

On the **Vault Setup** dialog, all pages that require a Vault connection are now removed or displayed based on sign-in status without the need to reopen the setup dialog.

Signed out, tabs have been hidden (except Sign In and Users):

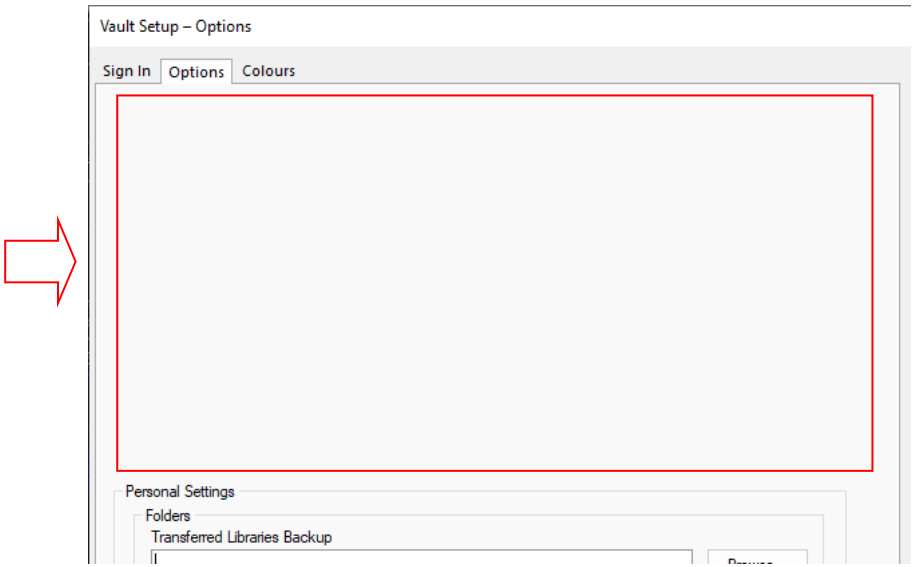


Signed in, all tabs are available:



Vault Setup – Visibility in Options page reflects the Sign-in status

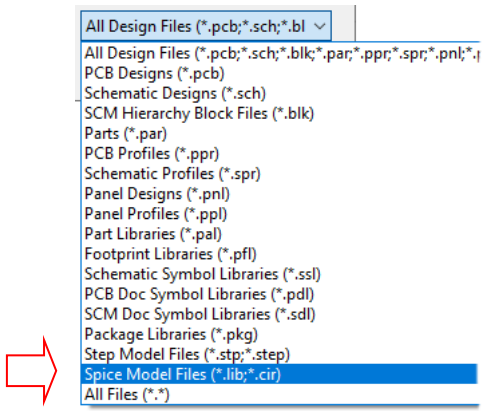
On the **Vault Setup** dialog, the **Options** page now reflects the sign-in status and hides information that would only be visible when signed in.



Spice Models available in Vault

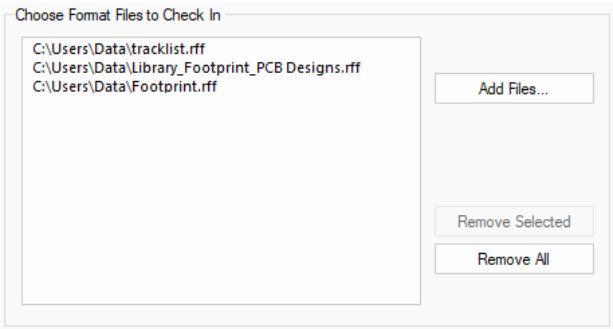
From within the **Vault Manager**, you can use the **Libraries To Vault** option to add the **Spice Model** data type to the Vault.

The Spice model data type is acceptable to the Vault. As well as **Import Vault Libraries** and **Libraries to Vault**, a Spice Model can also be checked out or copied out.



Format Files available in Vault

Report Maker Format Files can now be checked into to the Vault using the **Format Files to Vault** page in the **Vault Manager** dialog.



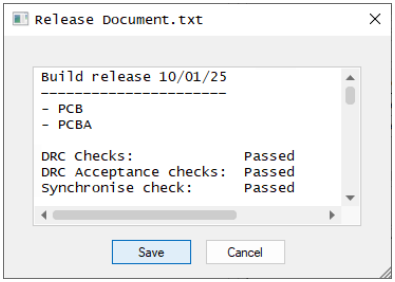
Once added to the Vault, format files can be managed through the Vault Browser page or directly from **Report Maker** dialog using the new **Vault** buttons. The Format Files folder and combo boxes can now handle Vault virtual folders and format files stored in the Vault.

Vault Browser – View Text Vault Files

View functionality is now available for vault **text** files.

From the **Vault Manager** and **Browse** page, select the text item and press the **View** button.

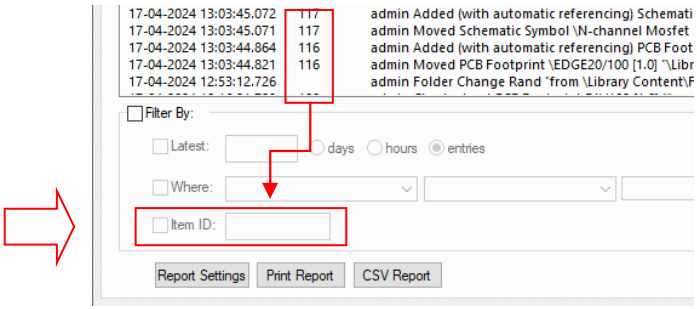
A dialog appears displaying the text in the file. There is also a **Save** button to save the text file locally.



This feature was back-fitted to V13.0.

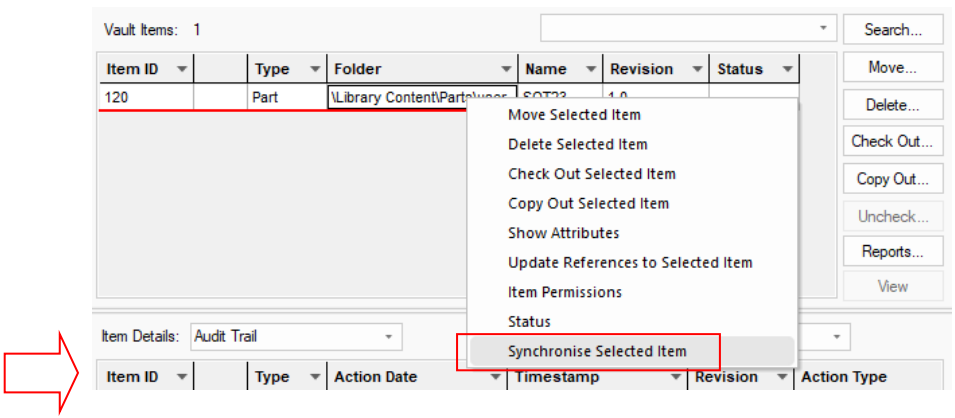
Vault Audit trail – Filter the Audit Trail on Item ID

You can now filter the Audit Trail based on a new filter check option **Item ID** This allows you to filter items in the audit trail list based on the item's ID.



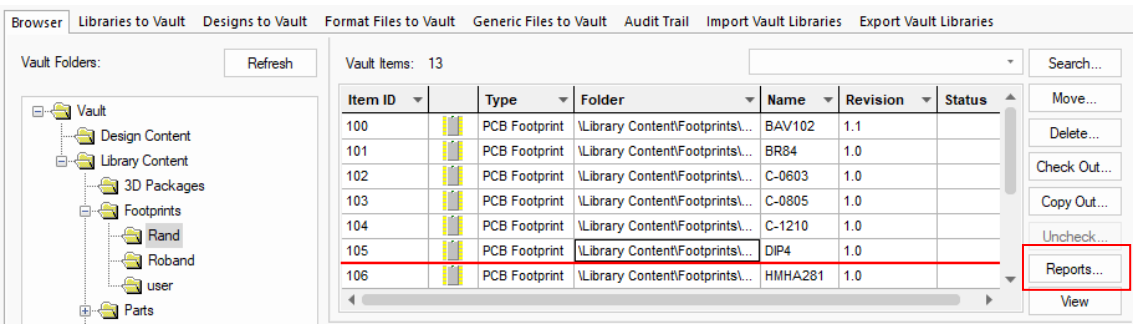
Vault Browser – Synchronise Selected Item

From the **Vault Browser**, once **Search** has been used, you can synchronise the **Vault Folders** tree to the selected item path. Right click on the selected item to choose the new option **Synchronise Selected Item** from the context menu.



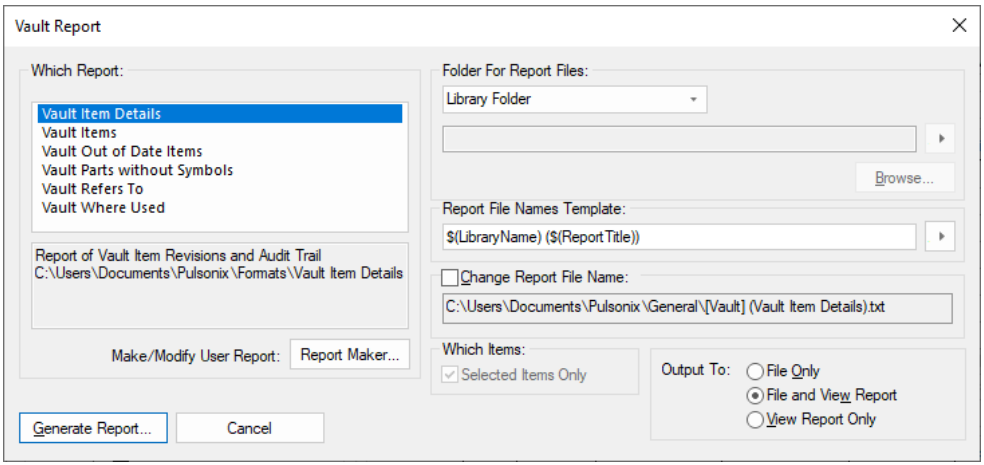
Vault Reports – Report Vault Parts without Symbols

A new report **Vault Parts without Symbols** is supplied with Pulsonix V14 and is available from the **Vault Manager**, **Browser** dialog and **Reports** button.



This report will list all Parts in the Vault that only have a PCB Footprint (PCB-Only Part) and Parts that only have a Schematic Symbol (Schematic-Only Part).

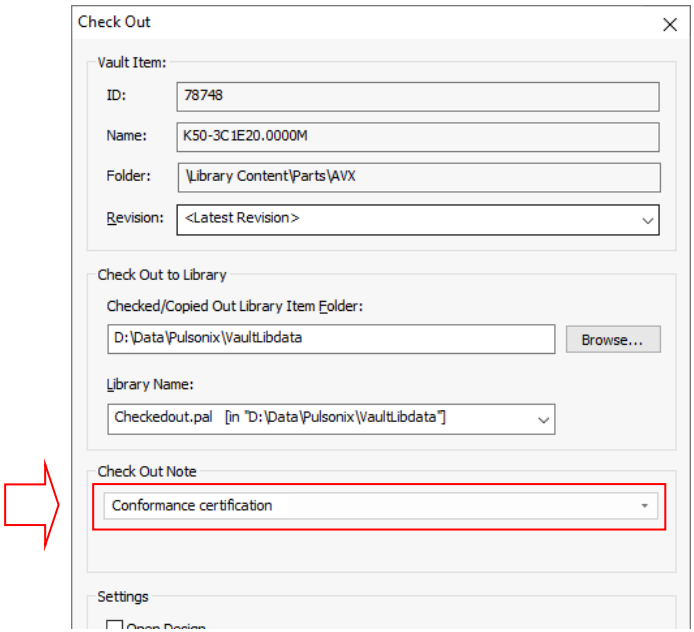
This report requires the connected Vault to have **Library Referencing** enabled.



Recent Notes option in Vault

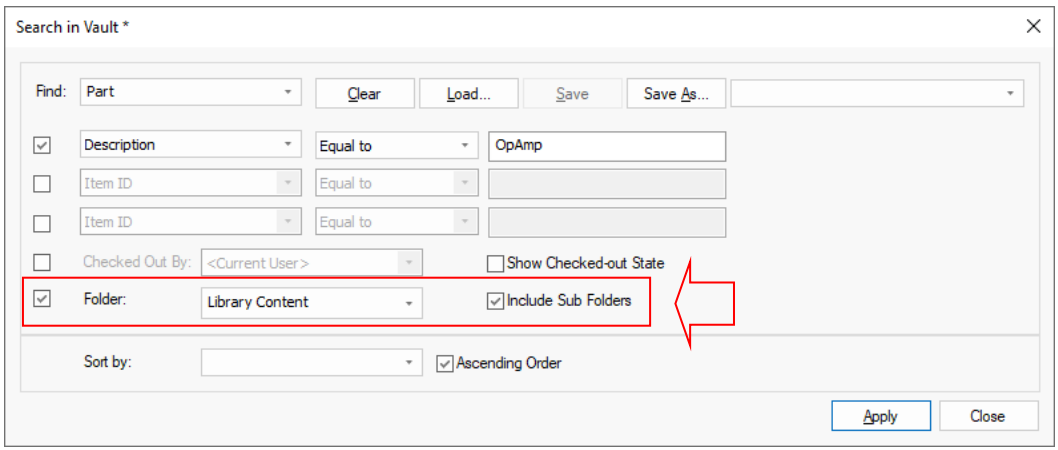
Check In, **Out** and **Uncheck notes** have been improved, and now are presented in a drop down list.

This box allows you to select a note that was last used by any Vault type or create a new one. Any changes to the note will be saved as a new note template for that Vault type.



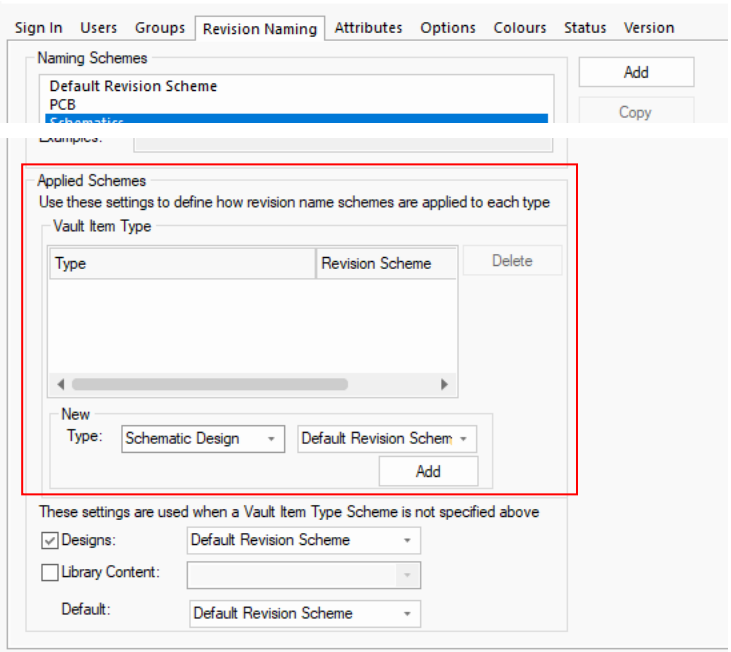
Vault Browser – Folder Search Filter

In the **Vault Browser Search** dialog, you can now filter search results by a virtual folder. There are a new set of controls in the dialog to displays all the virtual folders used in the Vault. When entering the dialog with no current search settings active, the folder selected in the virtual folder tree on the Vault Browser dialog will be pre-selected in the combo box. Alternatively, you can select a different folder. You can also optionally include sub-folders of the chosen folder by selecting the **Include Sub Folders** check box.



Type Specific Vault Revision Schemes

On the **Vault Setup** page under **Revision Naming**, you can now apply naming schemes to individual Vault item types.



This can be done by selecting a **Type** and a **Naming Scheme** from the drop down list boxes and clicking **Add** in the **New** section.

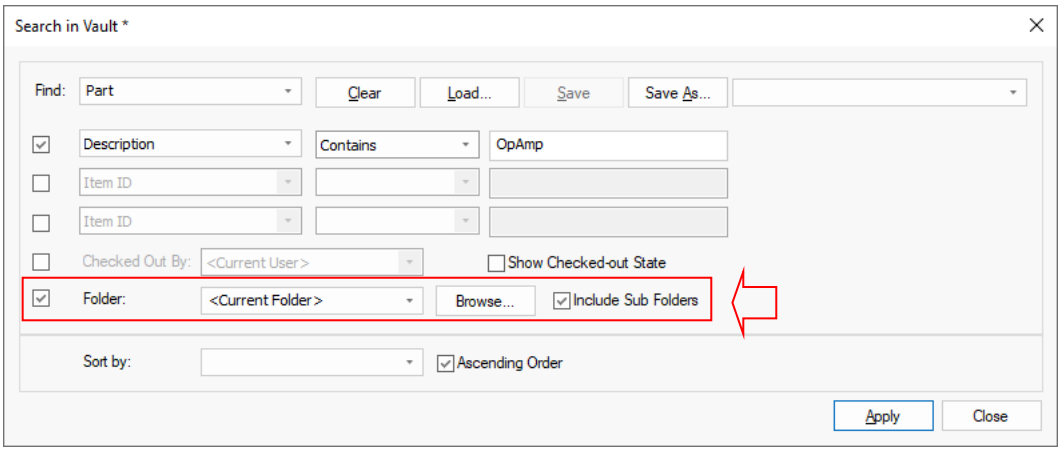
This will add that applied scheme to the list box in the **Vault Item Type** section.

You can also delete applied schemes using the **Delete** button to the right of the list box.

When checking in a new item, the revision scheme used will be obtained in the following order: Type specific scheme, Class specific scheme then Default scheme depending on the existence of the prioritised scheme.

Vault Browser – Folder Searching

From the Vault Browser, within the **Search** dialog, the **Folder** search now allows two options of filtering folders.



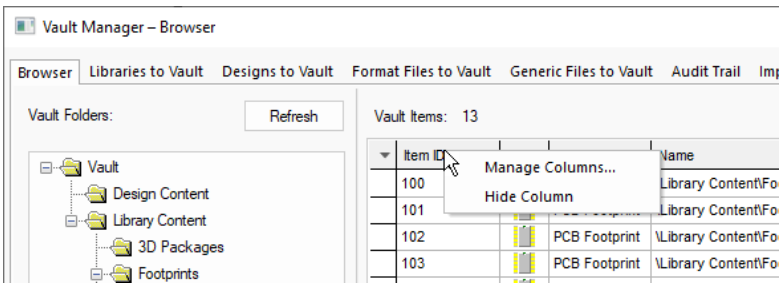
The **Folder** list box displays the last 10 folders used as well as the current folder.

You can also now use the **Browse…** button that opens the **Browse Virtual Folder** dialog enabling you can select a folder using the virtual folder tree.

Vault Attributes in Browser Grid

From the **Vault Manager** – **Browser** page, you can now customise the visibility of columns in the items grid. This includes adding new columns from a list of attributes associated with the Vault.

Opening the context menu by right clicking a column header:



This provides you with two options:

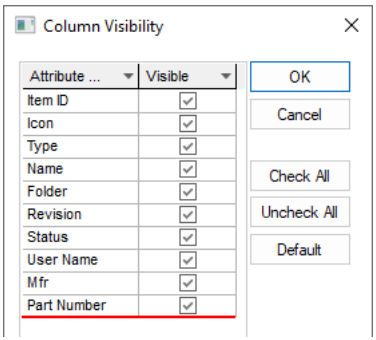
Manage Columns… - This opens the Column Visibility dialog where you can set the visibility of all columns.

Hide Column – This will hide the column where the right mouse button was clicked.

Column Visibility

Attribute Visibility Grid

The grid will show all the attributes that can be added the Vault Browser item grid. Attributes can be added using the Attributes page in the Vault Setup dialog. A tick in the check box next to the corresponding attribute indicates that this attribute should be visible.



Controls

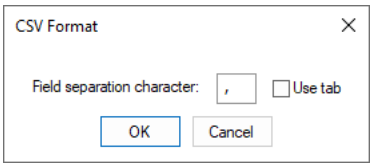
Check All - Will set the visibility of all attributes to true.

Uncheck All - Will set the visibility of all attributes to false.

Default - Will set the visibility of user attributes to false.

Vault Audit Trail Define CSV Separator For Report

From the **Vault Manager** – **Audit Trail** page, when exporting a **CSV Report**, a new dialog is shown that allows you to specify the CSV separator character. This works in the same fashion as the Attribute Editor CSV Export option.



Once the character is chosen and OK pressed, the report is written.

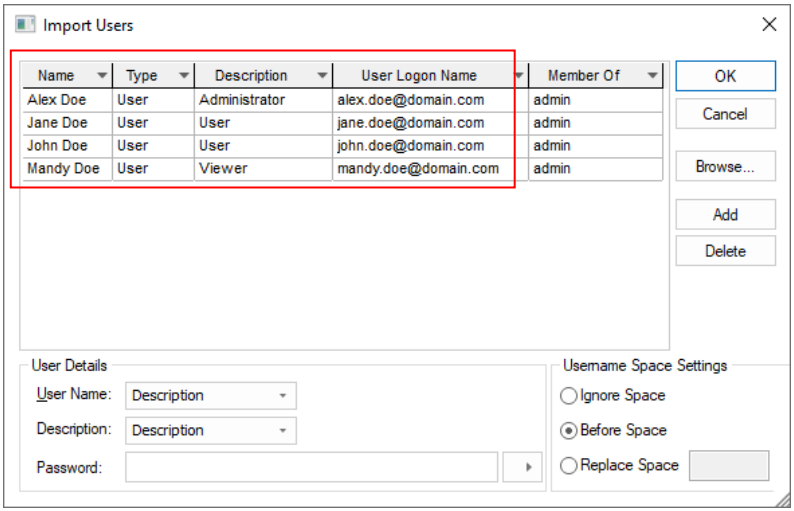
This feature was partially backfitted to V13.0. However, the dialog is not shown, but the CSV character can be set via the Attribute Editor CSV export.

Import Vault Users from CSV

You can now import users into the Vault from a CSV file such as a Windows Account list.

From the **Users** page of the **Vault Setup** dialog, there is a new button, **Import**.

When selected, this will open the **Import Users** dialog.



On this dialog, you can browse for the CSV file using the **Browse…** button. Once opened, the dialog grid will populate with the contents of the CSV file plus an extra column **Member Of**, which allows you to specify which **Vault group** the user should belong to. Using the selection in the **User Details** section, you can specify which column of the grid corresponds to the user's name and description.

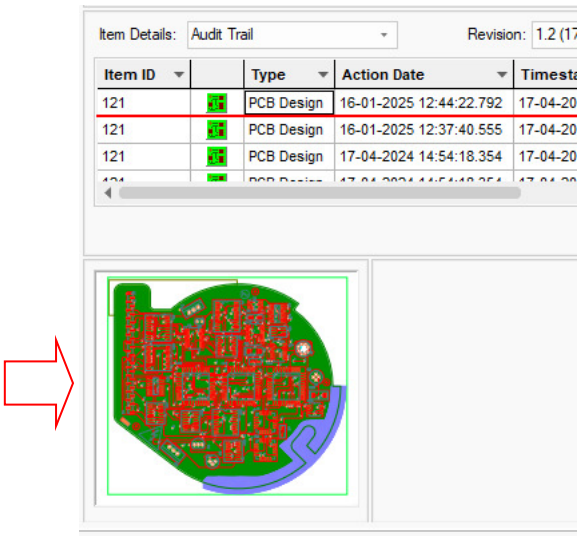
You can specify a password that is used for all the new users. Hard coded text can be added in addition to some built in tags (inserted by the right arrow button to the right of the password control) that allow for unique generation of characters.

The controls in the **Username Space Settings** can be used to control how spaces in the imported names are to be handled.

When 'OK' is pressed, the dialog will close and a report will be generated allowing you to review your changes before it is applied to the Vault. The report also shows you the passwords that will be applied to each user. Confirm changes by pressing Yes on the confirmation message box.

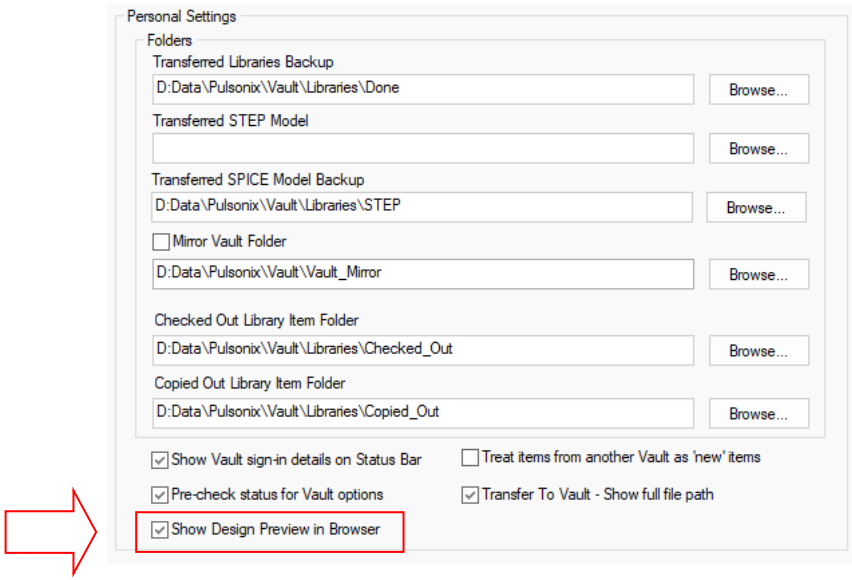
Vault Browser Shows Design Previews

The **Vault Browser** will now show previews of design-based Vault items such as PCB and Schematic designs.



For performance purposes, the preview uses a thumbnail image to avoid the need to fully draw the design each time it is selected in the browser. The thumbnail image is generated on first use and stored with the Vault data files.

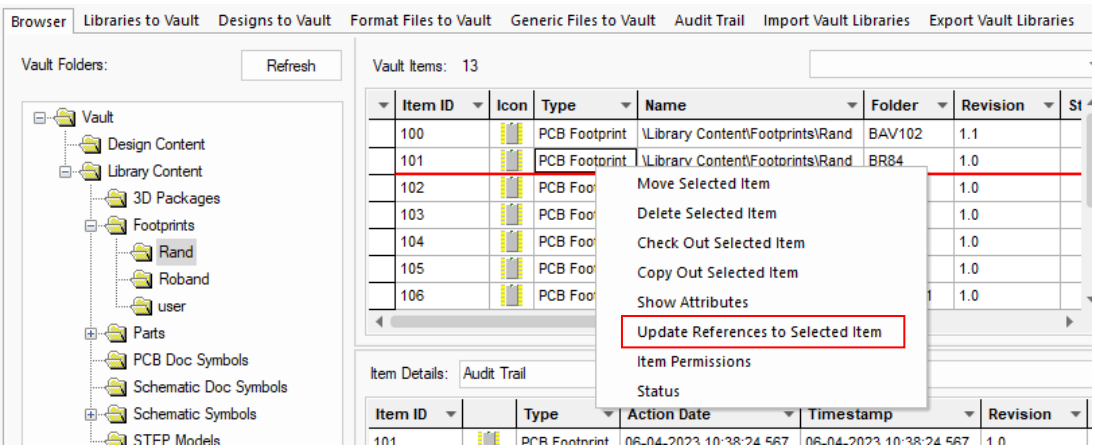
In the **Vault Setup, Options** dialog under **Personal Settings** there is a new option **Show Design Preview in Browser** option. By unchecking this option you can stop these previews from being shown which may be useful when you are accessing the Vault over a relatively slow connection such as a VPN.



Update Vault Cross-References Progress Dialog

The **Update Vault Cross-References** dialog (available from the context menu) now displays a progress dialog when **Update** is pressed. The progress dialog displays the name of each part that it's updating.

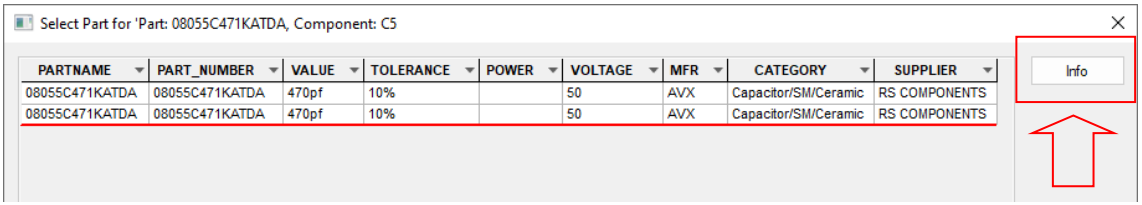
You can access this dialog through the **Vault Manager** dialog by right clicking on a PCB Footprint or Schematic Symbol from the item grid and selecting **Update References to Selected Item**.



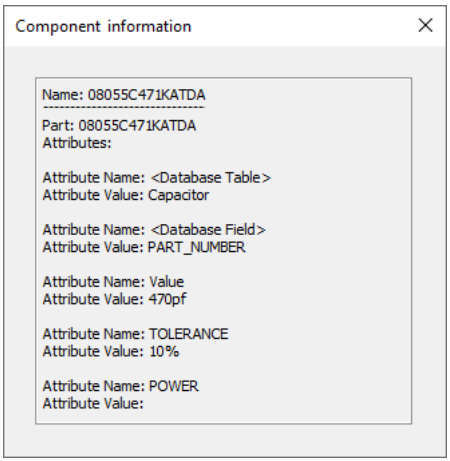
Pulsonix Database Connection (PDC) Option

Database Check and Update - Select Part dialog

When using the **Database Check and Update** option from the **Tools** menu, if **Update** is used, when the **Select Part** dialog is displayed there is now an additional button **Info**.



Pressing the **Info** button will display a report of information related to the Part currently being checked against the database, for example, Name, Part Name, Footprint, Attributes etc.



This feature was back-fitted to V13.0.

Overriding the Default Database Query Timeout

By default, all queries sent to the database will time out after a set period to avoid permanently blocking Pulsonix, causing it to freeze. This period will vary and will be defined by system parameters. Sometimes, particularly a with very large database, the time out period can be too short to allow the Database Connection to complete fetching the data it requires. In this case, it is desirable to be able to override the default time out period for Pulsonix without affecting the system wide default setting.

To achieve this, it is possible to manually edit the DCC file, using Windows Notepad for example, to add a Settings section that defines an alternative query timeout period. The Settings section should be added to the end of the DCC file immediately before the closing bracket as shown in the example below. For brevity, the detail of other sections is shown as '...'.

```
(Pulsonix
  (HEADER
    ...
  )
  (Contents
    ...
  )
  (Mappings
    ...
  )
  (Exclusions
    ...
  )
  (Settings
    (QueryTimeout 45)
  )
)
```

This feature was back-fitted to V13.0.

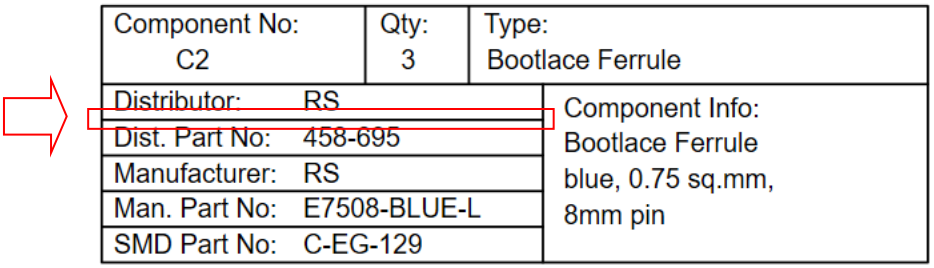
Report Maker Changes

Changes to Draw Line command

The **Draw Line** command now has two check boxes for Start Column and End Column.

Check the **Start Column** box to provide the column (as an order where 1 is the first) that the horizontal line will start from. Check the **End Column** box to provide the column that the line will draw to the end of. Unchecking either box will draw the line to the table extents.

For example, this allows you to draw a line under the first two columns only.

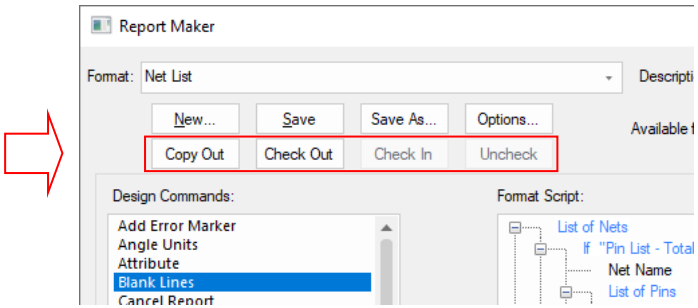


Check Out/In of Vault Format Files on Report Maker dialog

Report Format Format Files can now be added to the Vault. New format files can be added directly from the Report Maker dialog using the new Vault buttons on the Report Maker dialog (as well as the **Vault Manager** dialog and **Format Files to Vault** page).

If you have the Vault installed and are signed in, the Report Maker option will now allow you to **Copy Out**, **Check Out**, **Check In** and **Uncheck Report Maker** format Files.

Format Files folder and combo boxes can now handle Vault virtual folders and format files stored in the Vault.

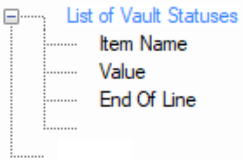


New commands to Report Vault Status

Report Maker has two new commands for reporting the vault status:

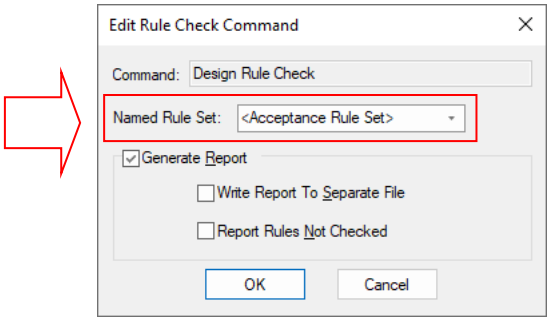
List of Vault Statuses - allows you to report all the Vault statuses that can be applied to items in the Vault. The command is available in Vault and Design contexts. Within this command, you can report the **Status Name** and its **Value**. The Value represents the Status position compared to other statuses in the Vault Status list.

Vault Status – allows you to report the Vault Status of a Vault item. This command is available for **List of Vault Items**. If the Vault item has a status, then the status name is reported, otherwise it will report nothing.



Design Rules Check and Electrical Rules Check command - Named Rule Sets

A named rule set list box has been added to the **Design Rules Check** and **Electrical Rules Check** commands. This will list any existing named rules found in the design.

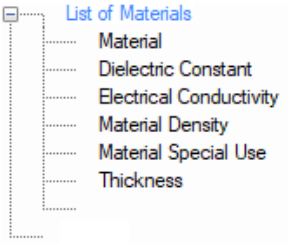


The default named rule set, <acceptance rule set> is used if you do no specify your own rule set.

New commands to support Material Parameters

New commands have been added to support the **Material** parameters defined in the **Technology** and **Materials** page.

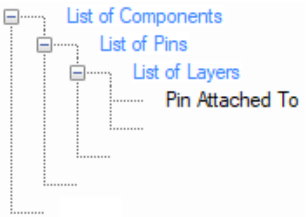
List of Materials command will report all fields for a Material.



Under the **List of Layers** command, you can report the **Material Density** and **Material Special Use** fields.

New command for List of Layers on Pin List with new 'Pin Attached To' field

When using the **List Of Pins** command, you can now go through a list of electrical layers from within the pin list using the **List of Layers** command.

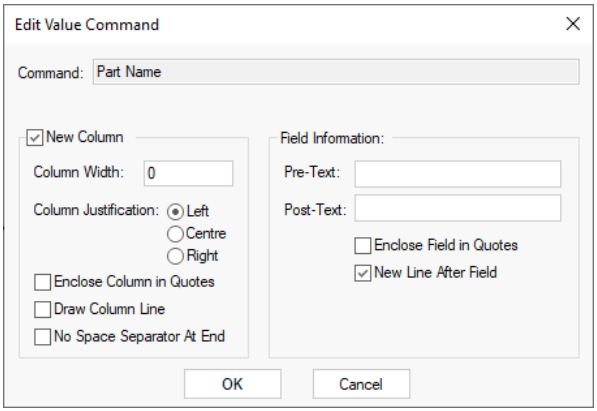


There is also a new field command **Pin Attached To** which will return the item the current pin is attached to on the current layer if it is attached on that layer. This new field command is only available from within a layer list if the parent of the layer list is a pin list at some point.

This feature was back-fitted to V13.0.

New command option to stop adding a space at the end of a column

In commands that report fields in a column now have an extra check box on their edit dialog - **No Space Separator At End**. When selected, this will stop automatically outputting a space between columns even if the Column Width is set to zero (0).

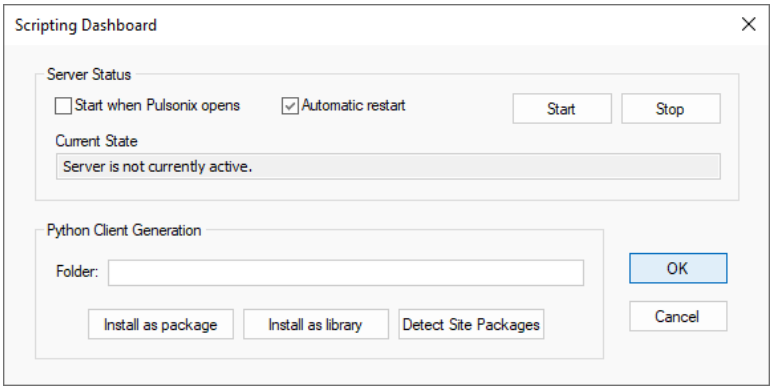


Major Update to Scripting – Support for Python 3.8

There is a new scripting interface to Pulsonix based on in-house technology. It is separate from ActiveX scripting and the API built on it, but can be used similarly to it using Python 3.

Currently, this will run along side the existing ActiveX scripting mechanism, but in time will supersede it.

The existing scripting mechanism has been renamed on the menu to **Run Script (Legacy)** and the new scripting mechanism will be call **Manage Scripting**.



Existing Scripts

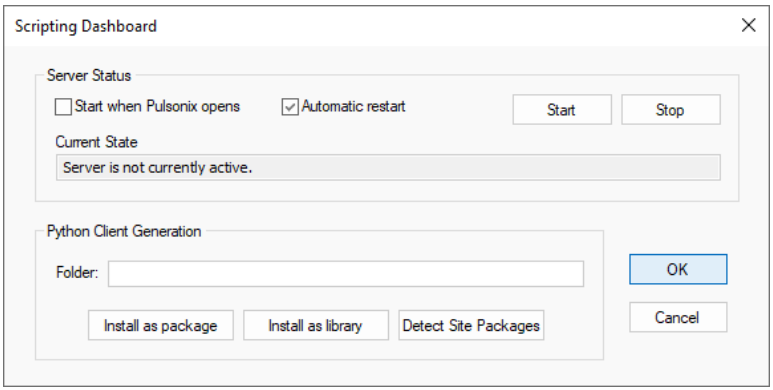
If you have scripts that have been created in the existing mechanism, they can still be used, or you can convert them to use the new mechanism. There are a few procedures to do to enable this:

- Install Python 3.10 or above
- We recommend you copy your existing script to another file as a backup before converting it. The file will be edited to accommodate the new scripting commands and will not be able to be run in the old mechanism once converted.
- You will now edit the existing script or its copy into the Python coding environment.
- Existing commands will need to be converted. The Python editor will highlight unsupported commands.
- Using the new environment and the new command structure is described in detail in the online help supplied with Pulsonix.

New Scripts using the Manage Script option

To use the new scripting mechanism and **Manage Scripting** option, you must have Python 3.10 or later installed first.

Once the Python environment is installed, the **Manage Scripting** option on the **Tools** menu opens the **Scripting Dashboard** from where the scripts can be controlled.



Scripts will be created within the Python environment.

Detailed information about using the new environment and the new command structure is described in the online help supplied with Pulsonix.

Using Python 3 scripts on another PC

If you create a script in Python 3.x that a colleague will also want to use, they must also have Python 3.10 or later installed on their machine.

Features included for the new Scripting Mechanism in V14

Below is a non-exhaustive list of features that have been added to the new scripting system and not available in the legacy system, (class names are shown in italics):

- Provided official Python 3.10+ support with the option to add support for other languages.
- Added *DesignShape* which provides access to line styles and filled status.
- Added functions on *Document* to create and delete line, pad and text styles.
- Added additional functions to *Material*.
- Added the ability to get *Board* types and names.
- Added more practical examples.
- Added the ability to get the *Symbol* of a *SymbolInstance*
- Added the ability to tell if a *CommonShape* is closed.
- Improved the interface to *Areas*.
- Improved the interface to Folders settings.
- Improved the interface to part and symbol libraries.
- Added the *PredefinedShape* class which is inherited by *PadStyle* and *PadStyleException*.
- Added new types of *PadStyles*

Scripting Pad Styles (Legacy Scripting) – New Objects - Plated and ForUseBy

Pad Style objects in scripting now have a **Plated** property, allowing you to get and set whether a pad style is plated (or not).

The **ForUseBy** method on a Pad Style object can now set and get the **Use By** state for the Through Mounting Hole and Surface Mounting Hole pad types.

This feature was back-fitted to V13.0 to the legacy scripting feature.

Pulsonix Sim – Spice Simulator

Using PulsonixSim

The PulsonixSim option is based on the NG Spice engine. This is a cost option and a new licence is required for this option.

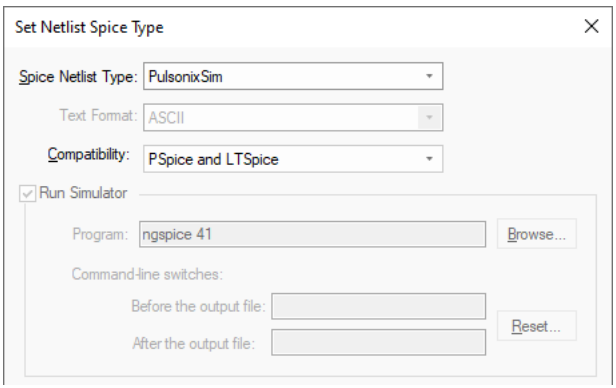
A reference is provided to the NGSpice Users Guide on our web site:

https://pulsonix.com/downloads/manuals/14.0/ngspice-44-manual.pdf

A link to this manual is also available under Help> Online Manuals> within Pulsonix itself.

Choosing the PulsonixSim Simulator

This simulator is enabled by selecting **PulsonixSim** as the **Spice Netlist Type**.



When switching to the PulsonixSim simulator, the **Simulation** menu changes to reflect this Spice functionality. The **LTSpice.pal** and **Spice.pal Parts** libraries will automatically be disabled for you on this mode selection.

A note about the switches for choosing LTSpice and PSpice compatibility, the default setting for PulsonixSim is for both to be defined, meaning that PulsonixSim is compatible with Spice modes defined for both simulator sources.

Libraries

PulsonixSim is supplied with its own Parts library called **PulsonixSim**

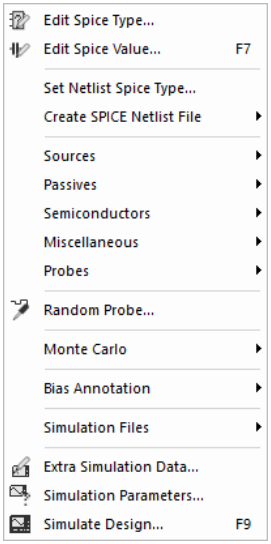
You should disable both the **LTSpice.pal** and **Spice.pal Parts** libraries under **Setup Folders** and **Libraries** so that these are not accidentally used.

The **PulsonixSim.pal** Parts library has been created to accommodate the nuances of the PulsonixSim simulator.

New Simulation Menu

The **Simulation** menu adjusts the contents depending on the **Spice Type** defined.

For PulsonixSim, the menu now looks like this:



The **Parts** toolbar is not available with PulsonixSim. However, related Parts, Sources, Probes and other design items are available on the simulation menu as their own sub-menus.

There is also a **Simulation** toolbar to use. This has commonly used **PulsonixSim** commands.

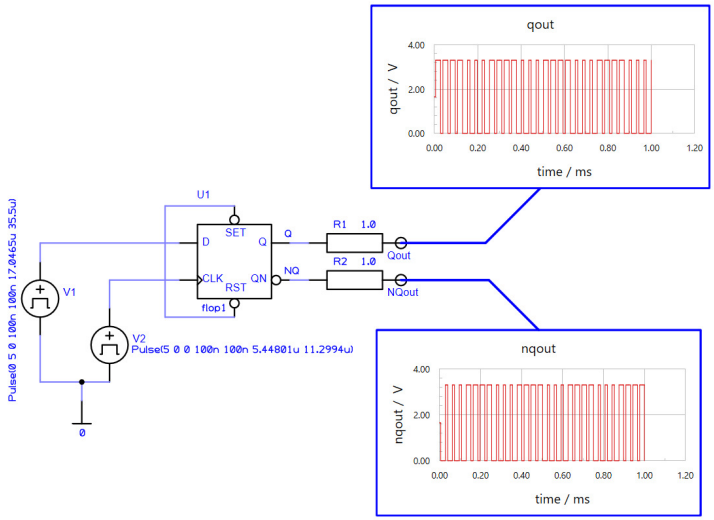
Using the **Simulation Files** sub-menu, you can view the circuit Netlist, the Log file of results when the simulator was run, including error reporting, warnings and general info on the simulation.

The Output driver file is the file used to tell the simulator what to do to the circuit.

Selecting any of these will display in your default text editor.

Adding Graphs into the design

The new simulator has full window graph capabilities and also local in-design Inserted Graphs.



Adding an Inserted Graph

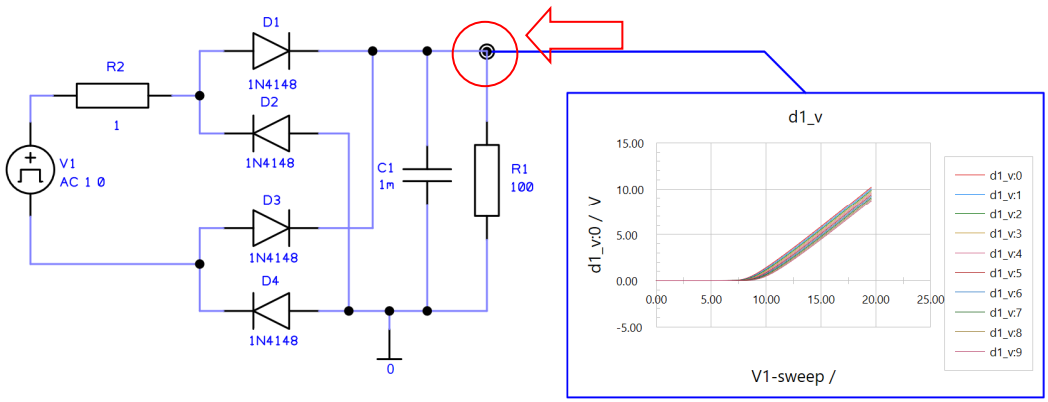
Add an Inserted Graph by selecting **Inserted Graph** from the **Simulation>** menu and **Probes>** . This option is also available on the **Insert** menu. A graph probe will be on the end of your cursor ready for positioning. Select a net to attached it to. Once attached, the graph window will now be available for positioning in your design. Once released, the graph will display simulation results. If no existing simulation results are available, run the simulator by pressing F9.

If you wish to use a full graph window to display the results, add Probes to the design.

To remove a graph window from the design, simply delete it using the **Delete** key.

Changing the Inserted Graph Probe

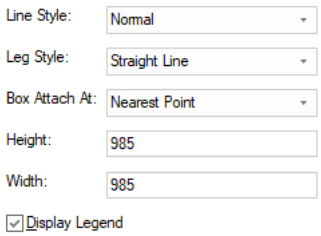
The probe used to attached the Inserted Graph to the connection is a **Part** named **Graph Probe**. This uses its own symbol and can be changed by you. The supplied Schematic symbol for this Part is **GProbe**.



Noe: If the Graph Probe component is not available, the Inserted Graph option will look for the ***Voltage Probe*** *Part in the library.*

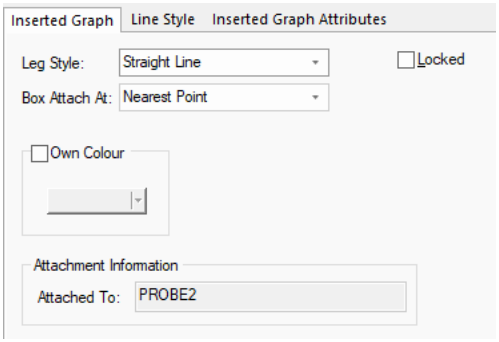
Default Design Setting Properties of an Inserted Graph

You can set basic default settings of an inserted graph window from the **Settings** menu**> Design Settings> Defaults> Inserted Graph** page:



Properties of an Inserted Graph

When an Inserted Graph has been inserted into the design, you can use **Properties** to change the graph shape line style and size, and the pointer line position. You can also change the **Line Style**, **attachment** location and its **own colour**:



Resizing an Inserted Graph

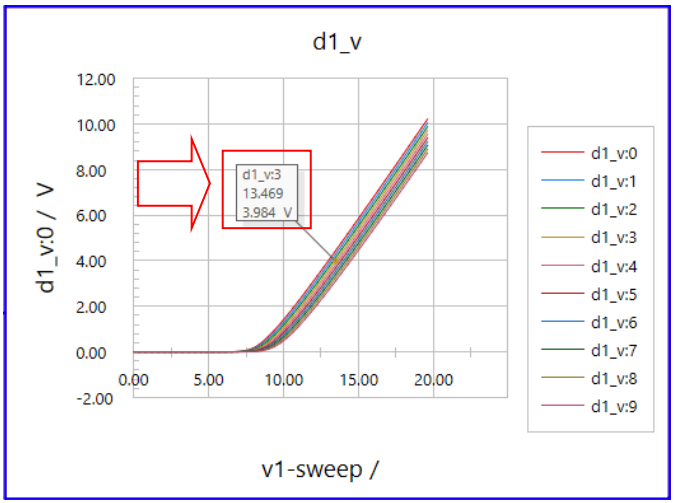
The Inserted Graph outline shape works the same as a regular shape. If you drag one side of the outline to resize it, when you release the mouse the shape releases. This way, you can make the window as big or small as required.

Moving an Inserted Graph

Pick on the framework around the graph **but not the shape outline**. Selecting the shape outline will edit the shape.

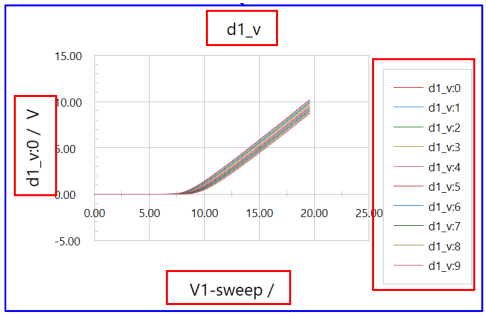
Graph Tooltip

If you click in the Inserted Graph a tooltip will be displayed. This shows you the net name, Time and Voltage at the point of the line.



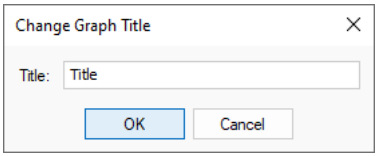
Graph Labels

If you select the labels within the graph windows, you get context dialogs that can be used to edit information within them.



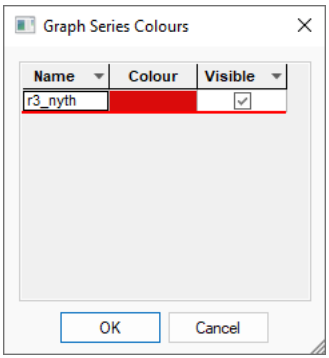
Title, Net Name, Time

Selecting the text of these labels displays a small dialog from where each label can be edited:



Graph Series

Selecting the bar or text to the right of the graph displays a small dialog from where the colour and display of signals can be defined:



Context menu options for an Inserted Graph

If you select one side of the Inserted Graph, you will see the 'shape' context menu and options related to shapes.

If you select the whole Inserted Graph shape or inside the shape on the graph, you can right click and select options from the context menu:

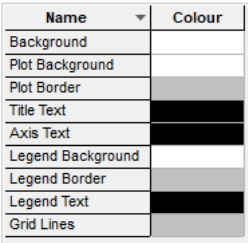
Export Sim Graph – export the graph to a bitmap format file.

Convert to Graph Window – converts from an in-design graph to a full graph window. This closes the Inserted Graph and opens a full graph window. It replaces the Inserted Graph probe with a graph probe.

Open Graph Window – switches and duplicates an in-design graph to a full graph window.

Changing the Colour of Graph Windows

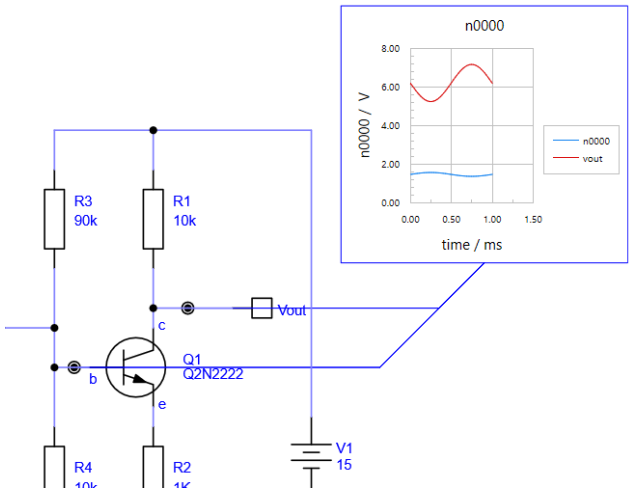
Colours for aspects of Inserted Graph can be changed within the **Colours** dialog and **Graphs** page.



Combining Inserted Graphs

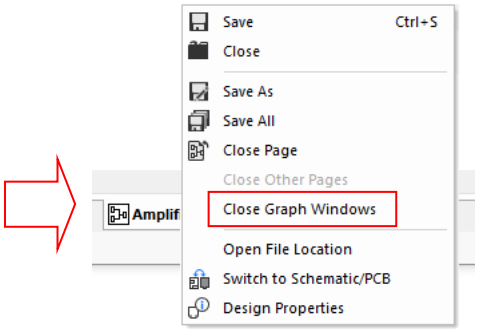
If you wish to overlay two or more signals into one Inserted Graph, drag an Inserted Graph over another one; the one you wish to combine it into. Click the 'host' graph to complete the combining. You will now see two (or more) lines directing from the originating nets.

The styling of the leader lines from the Inserted Graph shape will be the same now and defined in the shape **Properties**.



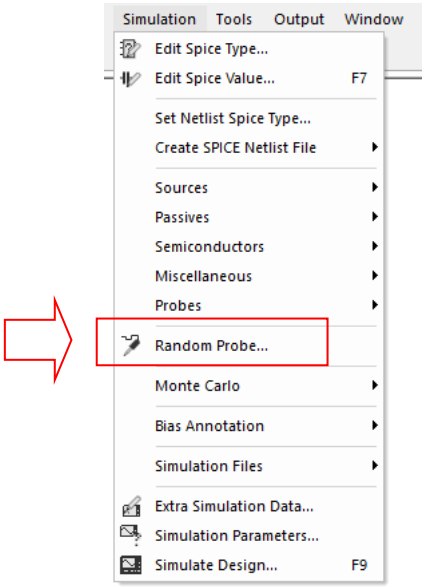
Closing a Full Graph Window

If running multiple full graph windows, you can close the graph(s) by right clicking on the master schematic workbook tab and selecting **Close Graph Windows**.

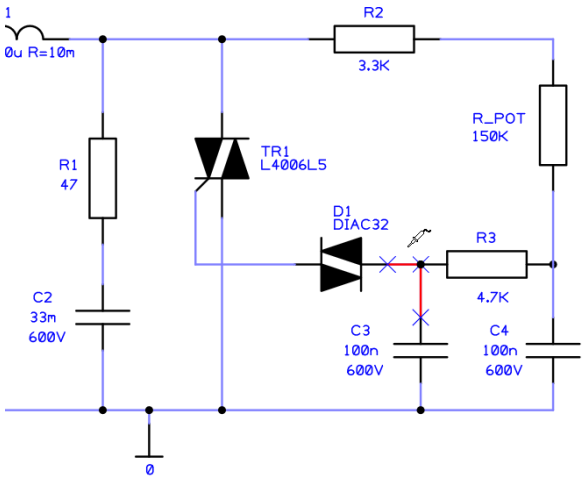


Random Probing

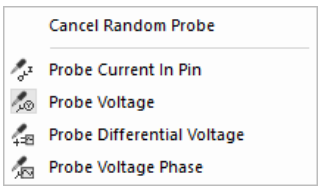
Once simulation has been run, you can choose to **Random Probe** from the **Simulation** menu:



Once selected, a **Random Probe** cursor is displayed:



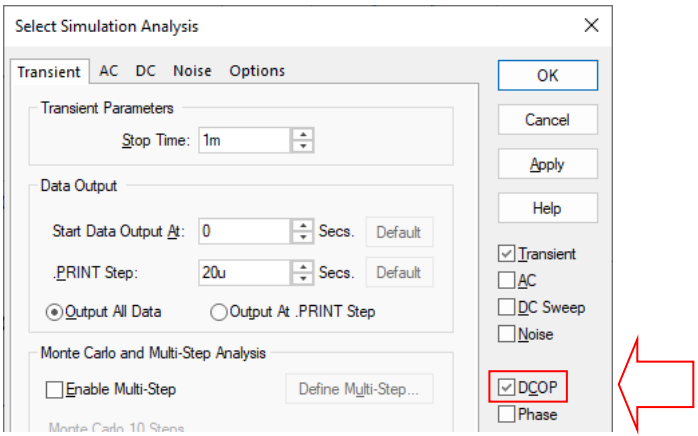
Once a Random Probe has been selected, you can right click and from the context menu, select the type of probe to use:



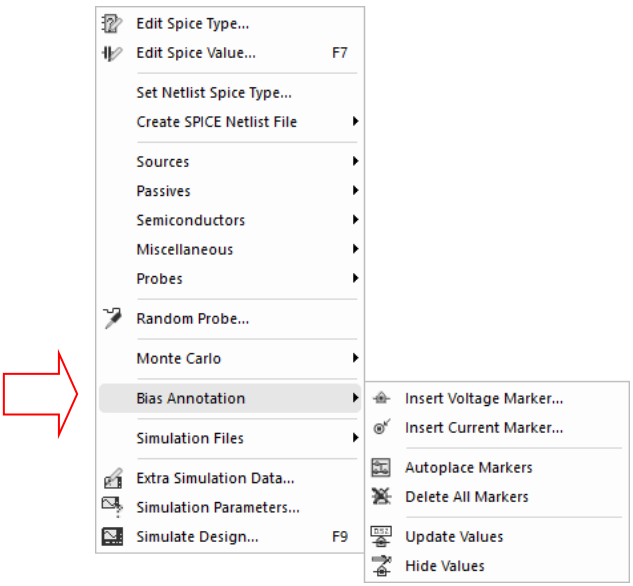
As you probe, a full graph window will be drawn. These are transient and not saved with the design and no 'probe' is inserted into the design. Effectively, these act the same way as if you probed a real design using a scope; you get the results at that moment. Each net probed will be displayed in the graph and updated with the next one probed.

Bias Annotation

You can insert **Bias Annotation** markers into your design when running a **DC Operating Point** (**DCOP**) analysis:



From the **Simulation>** menu, select Bias **Annotation>** select **Insert Voltage** or **Current Marker** to add markers selectively, or **Autoplace Markers** to place through out the design.

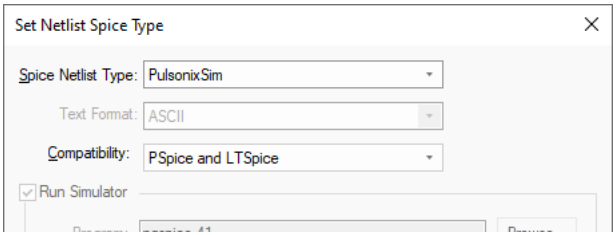


If using **Autoplace Markers** it will place Voltage Markers in the design.

Library Manager - SPICE Models Page

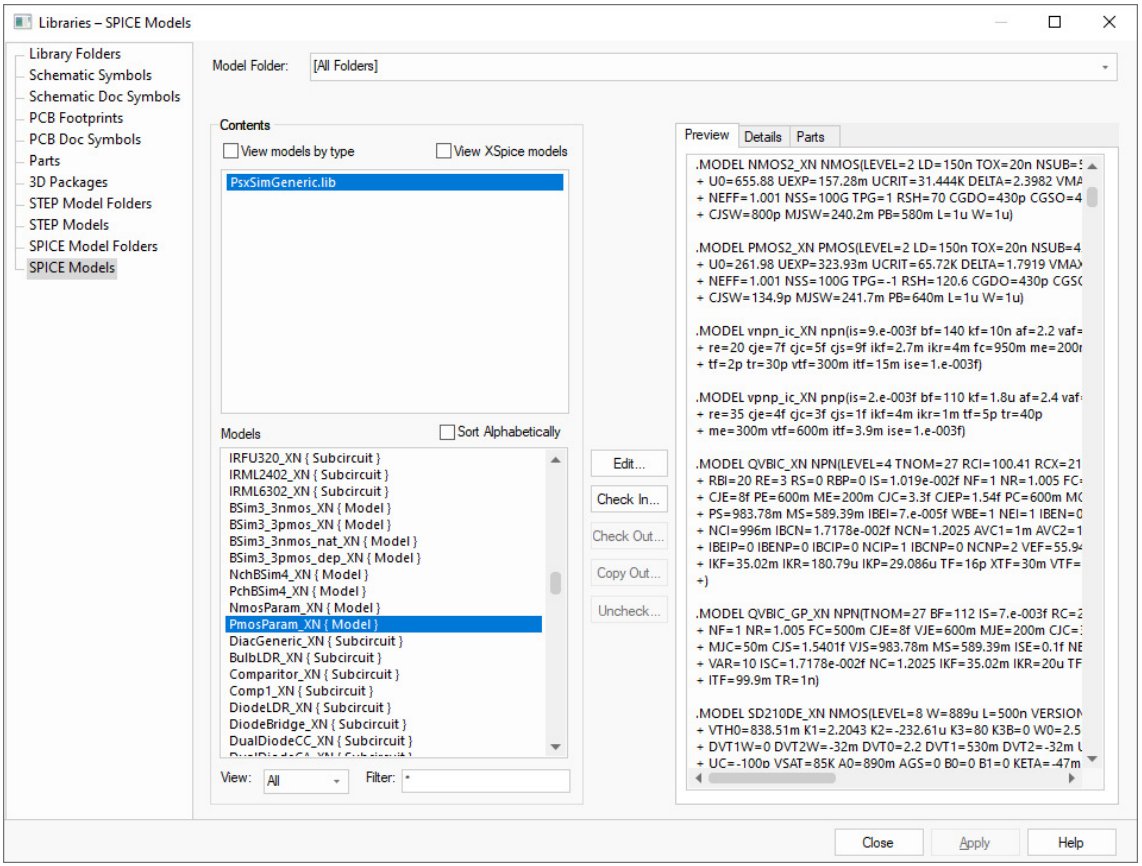
If you have the Pulsonix Sim Spice simulator licence, there are new **Spice Model Folder** and **Spice Models** pages available in the **Library Manager** from where you can edit and manage your Spice Models.

To see these pages in the **Library Manager**, you must also have your **Spice Netlist Type** set to **PulsonixSim** from the **Simulator** menu and **Set Netlist Spice Type**.



The **Spice Models Folder** page is a standard folder selection/management page, the same used for **STEP Models** and **Library Folders**.

The **Spice Models** page is used to manage your models allowing you to view and edit them.



Add part to design – SpiceModelsPage [NEW]

Antrea	08/05/2025	57647	No

New ‘add part to design’ button. Select a part from the ‘associated parts’ list and press the new button to add the part in the design. Parts from vault are also available.

Spice Value

The PulsonixSim engine uses the <Spice Value> attribute. For components using this, select the component and press F7 (Edit Spice Value).

Running PulsonixSim

Run the simulator by pressing **F9** or from the **Simulation** menu, **Simulate Design**.

Associated Simulation Files

Following a simulation run, a number of additional files are created and saved along side the design file. If you wish to move the design to another machine, and retain the simulation results, you should also move these files:

.cir - the Spice netlist of the design.

.out – the simulation results that are read by PulsonixSim to create the graph windows.

.log – this is a log of the last simulation run. This contains any errors or warnings produced when the last simulation has been run. This is not a critical file for simulation and is purely used for refence. It is not required if you move the design file.

.sch – this is the actual Pulsonix schematic design.

---

*Note: the Netlist, Output file and Log file can be viewed using the **Simulation Files** sub-menu from the main **Simulation** menu.*