Pulsonix Design System
V13.0 Update Notes

Copyright Notice

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 06/09/24

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

Version 13.0 Update Supplement

Installing the New Version of Pulsonix

It is always recommended you back-up all libraries, designs, technology files, profile files and report 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. You can install Pulsonix 13.0 over your existing V12.5 installation. If upgrading from V12.5 or an older version, you can install it alongside the older version if you prefer. In any case, you do not need to uninstall the old version first unless you wish to remove it from your hard drive.

64-bit Installation Folder

By default, Pulsonix V13.0 will be installed into the programs folder **C:\Program Files\Pulsonix13.0** and not C:\Program Files (X86).

Documentation Installation

The default installation locates all Pulsonix 'documents' (Master Libraries, Technology files etc.) is under user\documents\Pulsonix13.0 rather than being placed in public documents\PulsonixXX

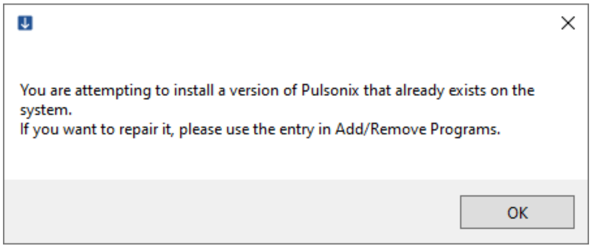
New Installer

Installer

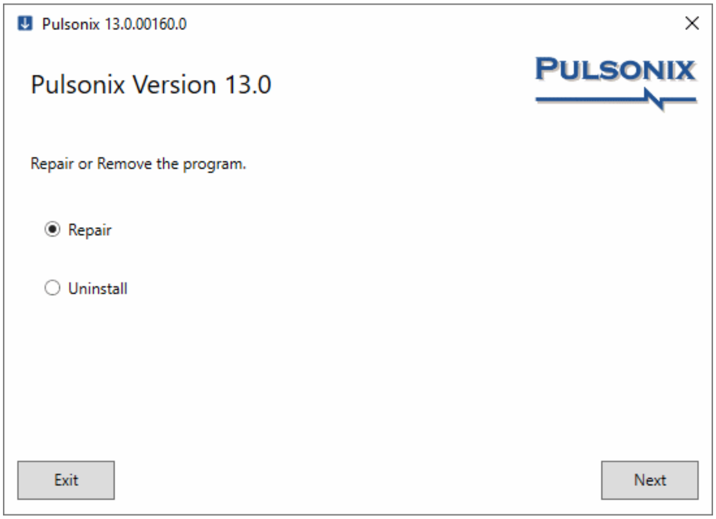
Pulsonix 13 has a new installer program. It still manages the installation, repair and uninstallation process as previous versions.

Repair or Uninstall

If you already have Pulsonix 13 installed and run the installer again, you will be instructed to run the add/repair option in the Settings dialog available on your Start menu.



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



Unattended Installation

In V12.5 and previous versions, there was a mechanism/procedure for doing an Unattended remote installation. In V13, this will be removed. This will be replaced with the MSI installer.

The **MSI installer** is available at the same location as the main Pulsonix installation set by using your Pulsonix login to our web site.

Licensing

Version 13.0 requires a new license if you are a new user or upgrading from any older version of Pulsonix earlier than and including V12.5. The new license would have been supplied to you under the terms of your maintenance contract.

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

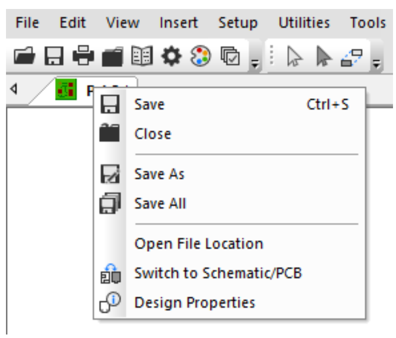
Version 13.0 Network License Server (NLS)

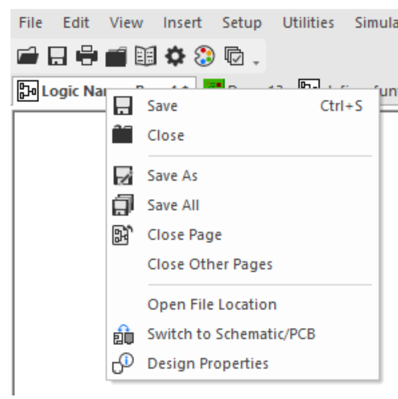
NLS has been updated for **Version 13.0** but an existing NLS installation of V12.5 will still run V13.0. However, in order to access new functionality and to take advantage of any issues fixed, you must install the new version of the NLS program.

New In Version 13.0

Workbook Tabs – Additional Context Menu Options

The workbook tabs context menu have been enhanced with the following options to allow quick access to useful options:





Save – An existing option

Close – An existing option

Close Page – An existing option (SCM)

Close Other Pages – An existing option (SCM)

New options:

Save As – Save the design with another name

Save All – For a PCB design, it saves every unsaved file currently open. For a Schematic with multiple sheets, it will save all the sheets as well

Open File Location – this will open the current design in your Windows Explorer

Save in Library – this is only available if the file being edited is from a library

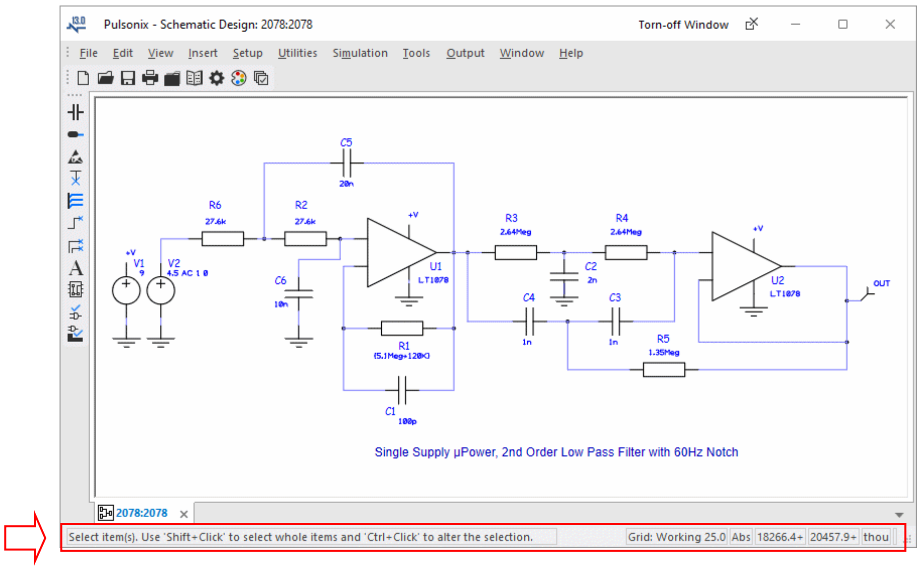
Switch Designs to Schematics/PCB – opens the equivalent 'opposite' design (with the same name) to the one open

Design Properties – will display the Design Properties dialog

New Features Added to Tear-off Windows

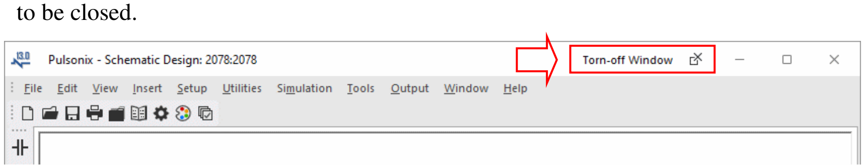
Status Bar

Each tear-off window now has its own unique Status bar. This will enable you to see actions and commands being performed in that window.



Tear-off Status on Caption Bar

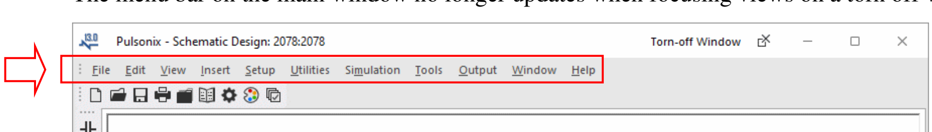
Each torn off window will now show the status with an additional new button to enable this window to be closed.



Menu Bars

Each torn-off window now has its own unique menu bar.

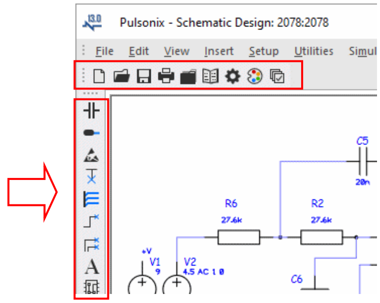
The menu bar on the main window no longer updates when focusing views on a torn off window.



Toolbars

Each torn off window now has its own set of toolbars.

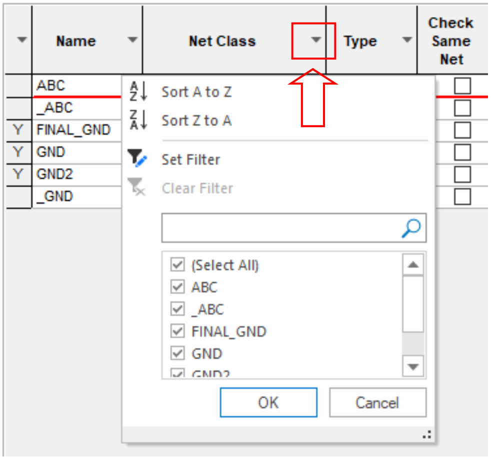
When a new torn-off window is created, a direct copy of the toolbars on the main window is made for the torn-off window's toolbars.



Note: Toolbar changes are only saved after Pulsonix is closed down, making a change and tearing off a new window doesn't show these changes immediately.

Filtering and Sorting of dialog Grids

Grid column headings now have a filter button that allows a filter to be set. When a filter is set, only rows with values matching the filter are visible. This feature is available on grids, such as within the **Technology** dialog and the **Grids** dialog. This feature is available in most grids but not all.



Clicking on the filter button (the small 'down' arrow next to the name header) displays a pop up menu with a list of commands:

- Sort A to Z - Sorts column rows in ascending order
- Sort Z to A - Sorts column rows in descending order
- Set Filter - Allows a custom wildcard filter to be set, using the Wildcard Wizard
- Clear Filter - Clears the currently set filter (enabled only when a filter is set)

- (Values list) - A check list box with search filter functionality that can be used to quickly filter row values

If a filter is set that matches none of the rows in the grid, the text **No items match your search** is shown.

Multiple columns can be filtered at the same time to further refine the filtered rows.

To show that a column has been filtered, a filter icon is shown on the filter button. If sorting has been applied, an arrow is also shown on the button to indicate the sorting direction (ascending or descending).

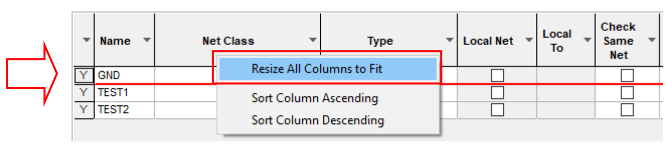
Hovering the mouse over the filter button in a column header will show a tooltip with the current filter information for that column.

Right-clicking on a column header when one or more filters are set will show a context menu with the command **Clear All Column Filtering**, which will remove all filters set on the columns in the grid.

The filter popup menu can be resized to allow more items in the values list to be shown.

Dialog Grids – 'Resize All Columns to Fit' Command

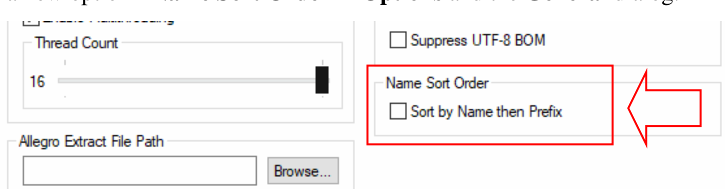
In dialog grids, all columns can now be resized to fit their contents by using the command **Resize All Columns to Fit**. This option is available on the context menu when right-clicking a column header.



Previously, only a single column at a time could be resized to fit its contents by double-clicking on the right edge of the column header.

Define Name Sort Order Feature

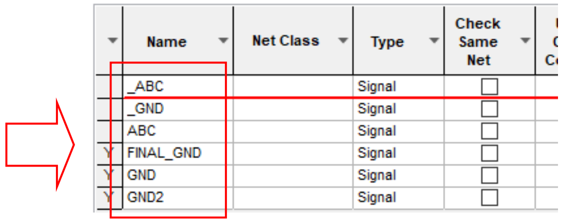
There is a new option - **Name Sort Order** in **Options** and the **General** dialog.



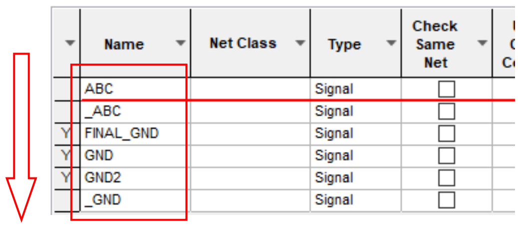
When selected, the **Sort by Name then Prefix** option causes the sorting of names to consider any alphanumeric characters before any prefix of non-alphanumeric characters. It will therefore group names by their alphanumeric portion first. For example: A, (A), _A, B, (B), _B rather than (A), (B), _A, _B, A, B. This is particularly useful when barring is used as the barred and unbarred strings are next to each other in a list.

Note: the sort order is applied to **all** sorted lists of names or strings, and including **Net Names**.

An example of unsorted Net Names, might look like this:



Once the sorting has been applied, it now looks like this:



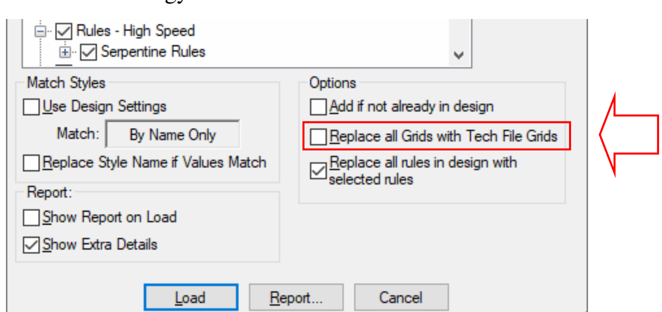
Default Pulsonix Theme

The default Pulsonix theme for new installations will now respect your current windows default app theme. This is noted here in case you notice it has changed.

Load Technology Dialog Changes

Replace All Grids

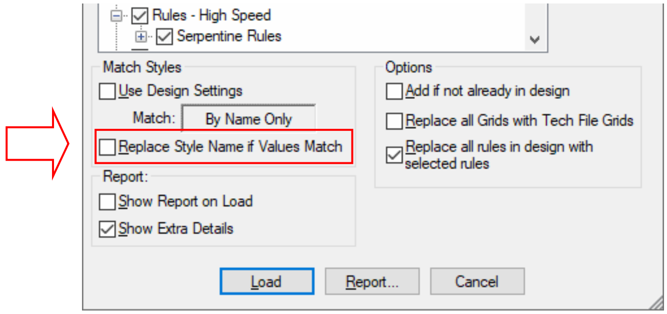
A new check box has been added to **Load Technology** dialog. When the **Replace all Grids with Tech File Grids** check box is selected, it will replace all the current **Grids** in the design with the ones in the selected Technology File.



If you already have Grids selected in the selection tree and this new switch selected, the Grid in the tree is reload first, then overridden/replaced using the new Grids from the Tech file (because of this switch setting).

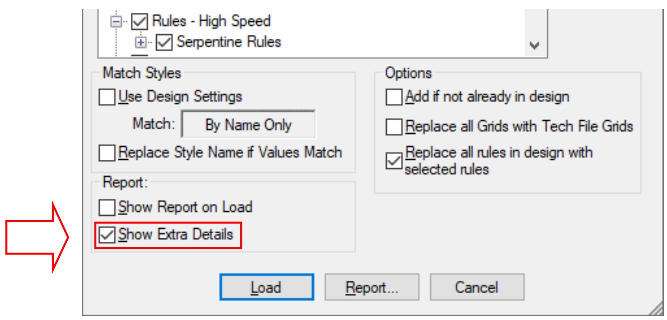
Replace Style Names

A new check box has been added to **Load Technology** dialog. When the **Replace Style Name if Values Match** check box is selected, Pad Styles in the Technology File that match their detail other than the name, will now change the name in the design technology instead of creating new styles. In other words if the name matches, then the design inherits the loaded style details from the Technology File.



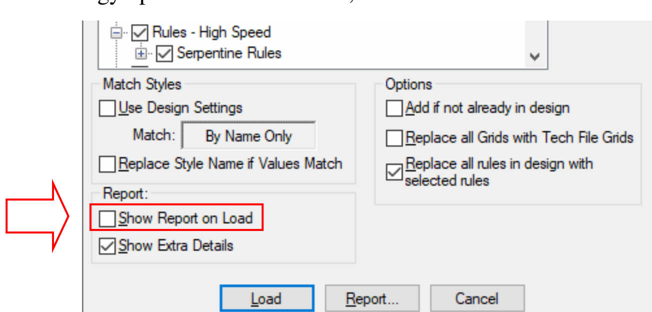
Report Detail Expansion

A new check box has been added to **Load Technology** dialog. When selected, the **Show Extra Details** check box will add more specific details about the changes that will be made when the technology file is loaded. For example, CAM/Plot settings changes.



Show/Suppress Report on Load

A new check box has been added to **Load Technology** dialog. When the **Show Report on Load** option is selected, the automatically generated report will be suppressed from view when the Load Technology option is run. However, it will still be created and saved.



Technology Dialog Changes

Component Variants Rules added

Within a **Schematic Technology** you can now define **Component Variant Rules**. This provides you with a fast mechanism for defining the **Fitted** and **Not-Fitted** status to items within **Components**, **Areas** and **Hierarchical Blocks** without defining each component individually.

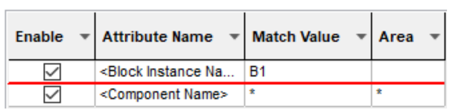
Your design must already contain a Variant(s) in order for this to be enabled.

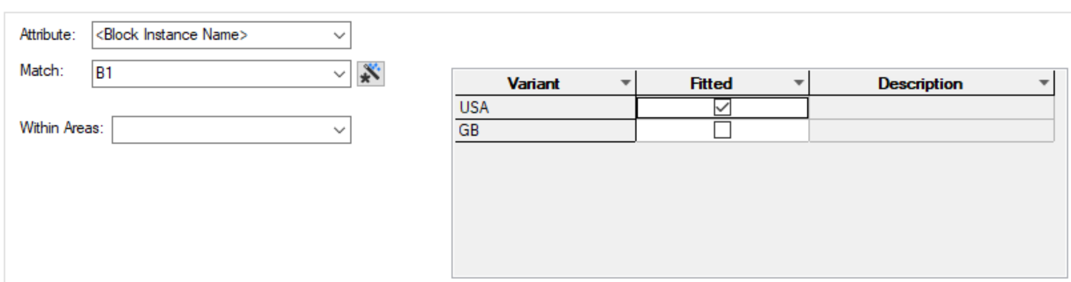
Any rules defined using these rules are passed through to the PCB.

If a fitted status is already exists on a Component, then the **Component Variant Rule** is ignored.

Component Variant Rules Dialog

The **Component Variant Rules** enables you to create the rule combination required along with the **Fitted** status (Fitted or Not-Fitted).





Technology pages – Rules columns can now be hidden

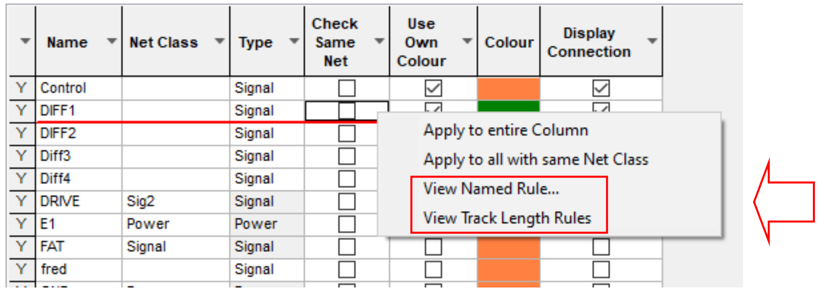
Two new buttons have been added to the context menu on some **Technology** pages to enable you to **View** or hide **Named Rule** and **View** or hide **Track Length Rule** (these were previously always displayed). The default state of these modes is off (rules columns are hidden with the button not checked). For example, within the **Net Names** dialog.

A new pair of 'Named Rule' columns has been added to the Nets pages in Technology. These columns allow the same rules modification as the existing rules columns (e.g. Track Length Rule, Differential Pair Gap Rule), but allow you to select the displayed rule.

To show or hide the Named Rule columns, right-click on a cell in the grid and select the 'View Named Rule' context menu command, which will have a check mark on it to show if the Named Rule is currently shown. This will show the 'Choose Named Rule' dialog which contains two controls: a check box to show/hide the Named Rule, and a combo box to select which rule to display.

The affected pages are: **Differential Pairs, Net Classes, Net Names, Signal Paths and Sub Nets.**

When in one of these pages, right clicking in the grid will display these options on the context menu:

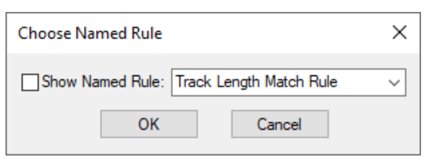


Settings for the Named Rule is saved for each page, so the visibility and selected rule will be remembered when closing and opening the Technology dialog. The default visibility for the Named Rule columns is hidden, and the default rule is first available rule alphabetically (e.g. Anti Pad Rule).

The **Named Rule** columns supports adding, editing and removing rules, in functionally the same way as the Rules list at the bottom of the page. These commands are shown on the context menu when right-clicking on a cell in the Named Rule column.

Note: Track Parallel Segments, Adjacent Nets, and Creepage rules cannot be used for the Named Rule because they require an additional match attribute and value that cannot be set from just the single Attribute and Match columns of the Named Rule.

When the **Named Rule** option is selected, a dialog is displayed. From here, you can choose the rule name to view.

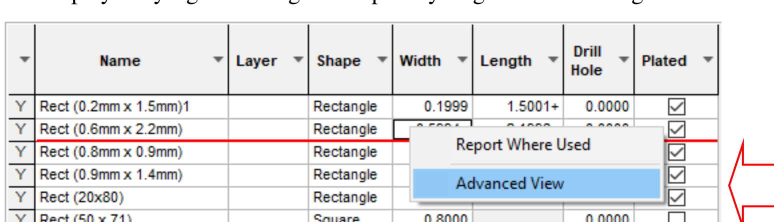


The **Show Named Rule** check box enables the rule to be viewed.

Pad Styles dialog – 'For Use By' Pad Types now shown in grid

In the **Technology Pad Styles** page, the **For Use By** pad types can now be shown as columns in the grid for layered designs. This means it is now possible to view and edit the used pad types for all pad styles simultaneously by right clicking on the check box that you require the status of and using the **Apply to entire Column** option.

This grid is displayed by right clicking on the pad styles grid and selecting **Advanced View**:



The full grid is then displayed with the **For Use By** settings displayed:

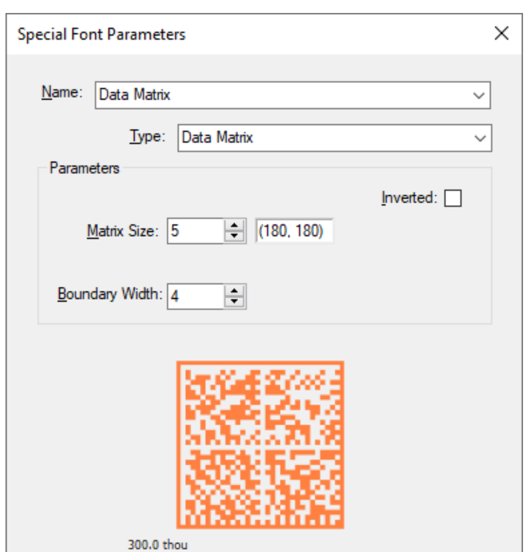
								For Use By					
	Name	Layer	Shape	Width	Length	Drill Hole	Plated	Through Hole Pads	Surface Mount Pads	Through Mounting Holes	Surface Mounting Holes	Vias	Micro-vias
Y	Rect (0.6mm x 2.2mm)		Rectangle	0.5994+	2.1996+	0.0000	☑	☑	☑	☑	☑	☑	☑
Y	Rect (0.8mm x 0.9mm)		Rectangle	0.8000	0.9000	0.0000	☑	☑	☑	☑	☑	☑	☑
Y	Rect (0.9mm x 1.4mm)		Rectangle	0.9000	1.4000	0.0000	☑	☑	☑	☑	☑	☑	☑
Y	Rect (2.0mm x 2.8mm)		Rectangle	2.0000	2.8000	0.0000	☑	☑	☑	☑	☑	☑	☑
Y	Rect (20x80)		Rectangle	0.5080	2.0320	0.0000	☑	☑	☑	☑	☑	☑	☑
Y	Rectangle (0.8mm x 0.9mm)		Rectangle	0.8000	0.9000	0.0000	☑	☑	☑	☑	☑	☑	☑
Y	Rectangle (0.9mm x 1.4mm)		Rectangle	0.8500	1.4000	0.0000	☑	☑	☑	☑	☑	☑	☑
Y	Rectangle13		Rectangle	0.3048	0.6604	0.0000	☐	☑	☑	☑	☑	☑	☑
Y	Rectangle14		Rectangle	0.8636	1.4986	0.0000	☐	☑	☑	☑	☑	☑	☑
Y	Round (0.4mm)		Round	0.4000		0.8750	☑	☑	☑	☑	☑	☑	☑
y	Via (18)		Round	0.4572		0.2540	☑	☑	☑	☑	☑	☑	☑
Y	Via (40)		Round	1.0160		0.7112	☑	☑	☑	☑	☑	☑	☑
Y	Via (40) Plugged		Round	1.0160		0.7112	☑	☑	☐	☐	☐	☑	☐
y	Via (41)		Round	1.0160		0.6096	☑	☑	☑	☑	☑	☑	☑
y	Via (50)		Round	1.2700		0.7620	☑	☑	☑	☑	☑	☑	☑
Y	Via (60)		Round	1.5240		0.8128	☑	☑	☑	☑	☑	☑	☑
y	Via 400 120		Round	0.4000		0.1200	☑	☐	☐	☐	☐	☐	☑
y	Via 500		Round	0.5000		0.2000	☑	☑	☑	☑	☑	☑	☑

Changes to Component Rules

A small change has been made to the way in which **Schematic Component Rules** are applied to **Blocks**. The colour defined now propagates down through all nested levels within the block. Previously, it only applied to the first level of block.

Data Matrix Text Font added

A new special font type of **Data Matrix** has been added to **Text Styles** in the **Technology** dialog. This allows you to create a special font type of **Data Matrix**. This is in addition to Barcodes and QR Code text fonts.



Matrix Size - Use a value between 1 and 15 to define the size of the Data Matrix code. The actual size in pixels is displayed next to the Matrix Size entry.

Boundary Width - The width in squares surrounding the Data Matrix code.

Inverted - Inverts the Data Matrix code.

Layer Span usage added

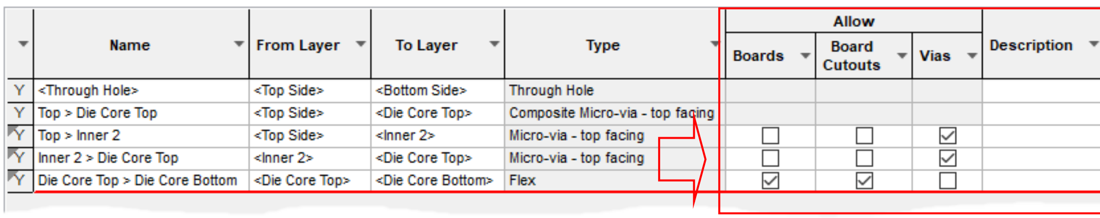
Within a **PCB Technology** you can now define the **Layer Span usage**. For **non-through hole** and **non-Composite** spans, you can define how the span will be used under **Allow**.

Selections are available for **Boards**, **Board Cutout** Areas (Cavities) and **Vias**.

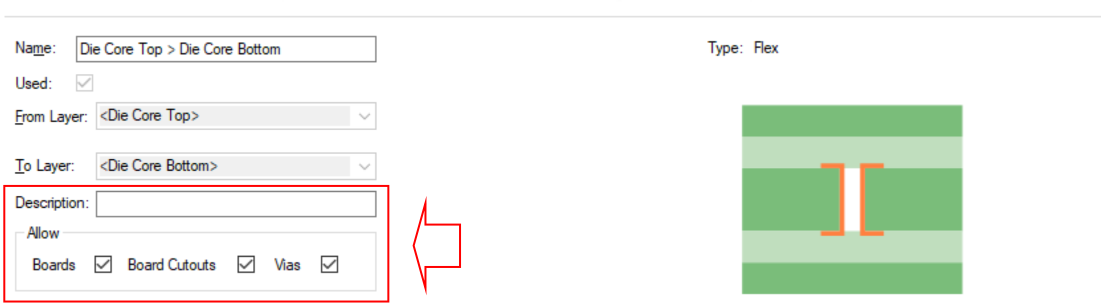
When using these items in your design, the usage allowed will be assessed (if defined) and the spans restricted to just the allowed type. This will help you refine your selection.

The Layer Spans dialog also enables you to add an optional **Description** for the span.

The new **'Allow'** fields and the **Description** can be exported using the **Report Maker** option (see below).

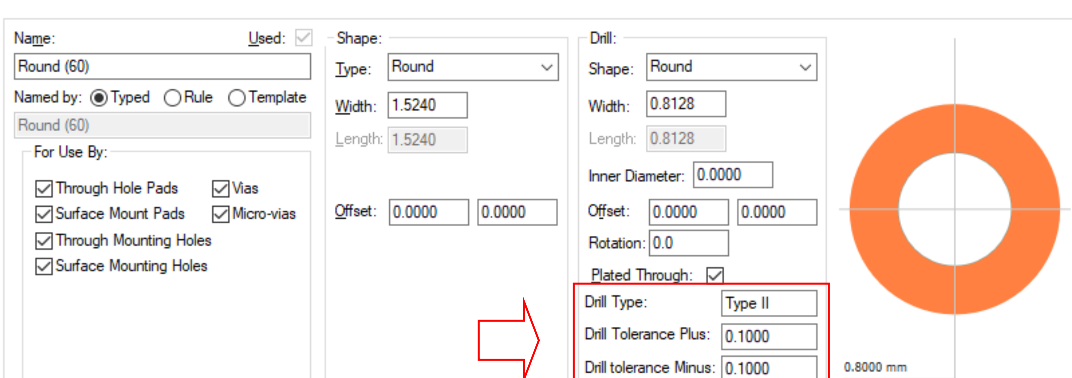


	Name	From Layer	To Layer	Type	Allow			Description
					Boards	Board Cutouts	Vias	
Y	<Through Hole>	<Top Side>	<Bottom Side>	Through Hole				
Y	Top > Die Core Top	<Top Side>	<Die Core Top>	Composite Micro-via - top facing				
Y	Top > Inner 2	<Top Side>	<Inner 2>	Micro-via - top facing	☐	☐	☑	
Y	Inner 2 > Die Core Top	<Inner 2>	<Die Core Top>	Micro-via - top facing	☐	☐	☑	
Y	Die Core Top > Die Core Bottom	<Die Core Top>	<Die Core Bottom>	Flex	☑	☑	☐	



Pad Styles has optional Drill Type & Tolerances

The drill on a pad style now has additional values - **Drill Type**, **Drill Tolerance Plus** and **Drill Tolerance Minus**.



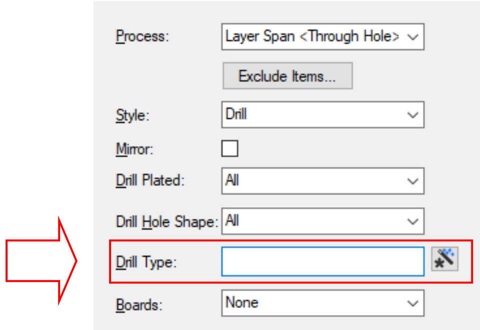
The **Drill Type** defines the drill as a special type which is separated from normal drill holes of the same size. This is a text field that you can type into. It would usually be a standard name, such as

Type II. Typically, a Drill Type would be specified for filled & capped vias. Once a Drill Type is defined, you can add **Drill Tolerances** of **Plus** and **Minus.**

Plotting Special Drill Types

The CAM Plot post processing of drills can optionally define a **Drill Type** match string using a wildcard string which will filter these special drill sizes so they can be outputted in a separate file.

When defining the drill output in the **CAM Plot Wizard**, the **Drill Type** is now available:



Drill Tables and Special Drill Types

When inserting Drill tables it separates the types but complex drill tables should be generated using the **Report Maker** and the table inserted into the design using the **Insert User Report** option.

Report Maker and Special Drill Types

The **Report Maker** option can now export the drill types and tolerances under the **List of Drill Sizes** and **List of Drill Holes** commands. See further down these Update Notes under Report Maker Changes.

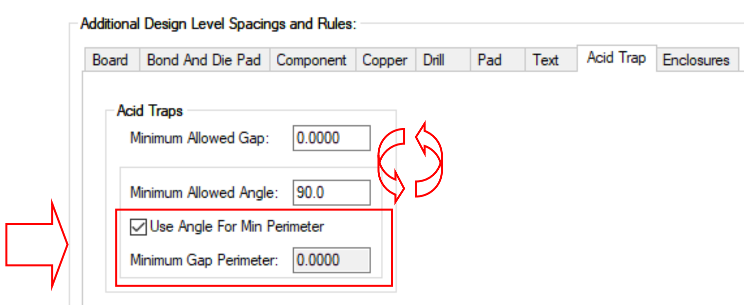
Other Outputs

Other outputs, such as ODB++ and IPC-2581 will also include the Drill Types and their values.

Changes to Acid Trap Rules and Checks

The **Track** tab under **Spacing Rules**, **Design Level** in the **Technology** has been renamed to be more reflective of the expanded checks that have been added. The tab is now named **Acid Traps**.

The page has already been rearranged to reflect additional parameters available and their association.



New commands have been added to this tab; **Use angle for Min Perimeter** and **Minimum Gap Perimeter**. The existing parameter, **Minimum Allowed Angle** has been moved down and grouped with the two new parameters as it now has a direct impact when the **Use angle for Min Perimeter** is enabled.

Use angle for Min Perimeter would be selected if you wish the system to calculate the Minimum Gap Perimeter based on the Minimum Allowed Angle.

Minimum Gap Perimeter is calculated if **Use Angle For Min Perimeter** is selected. With this not selected, you can type the perimeter value that you choose.

Note: If using an external program such as Valor for post design DFM checking, this parameter should be directly taken from the value defined in the Valor system.

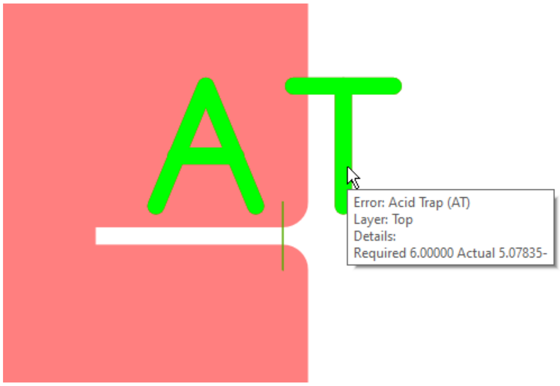
Details about this new functionality is described more in detail below.

Acid Trap Check Overhaul

The acid trap check under **DRC**, **Manufacturing**, **Acid Traps** has been overhauled. The updated DRC checks are a direct reflection of the **Acid Trap** changes in the **Technology** dialog under **Spacings Design Level** and **Acid Traps** (see above section).

The previous functionality of checking angles that were too sharp is still available, but now it also checks for nooks and corners where acid may get trapped in a piece of copper. There is no interface change to the DRC dialog, just functional changes underneath.

The two examples below show a small gap in the copper shape:

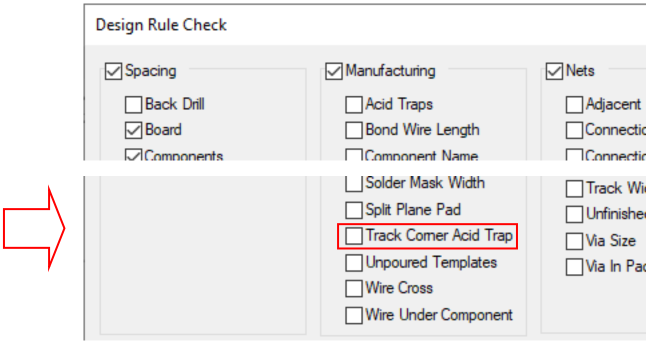


And a small gap created on the same copper shape.

Track Corner Acid Trap DRC Check

With changes made to the **Copper Check Rules** and the introduction of new **Acid Trap** rules and checks, the existing **Acid Track check** has been retained as a quick check, and less detailed.

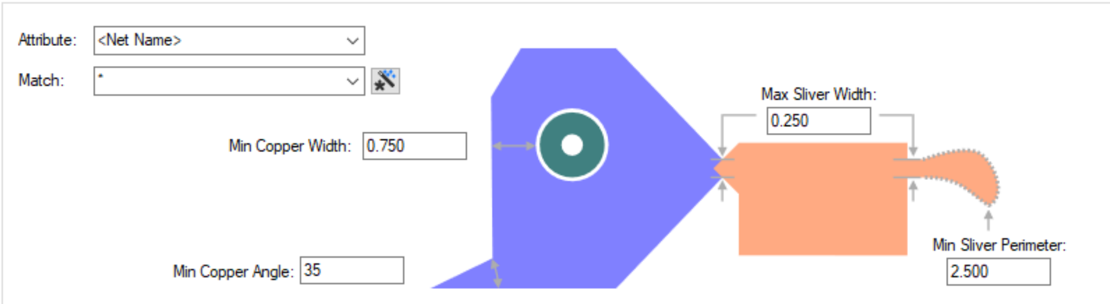
As such, the current version of the Acid Trap check has been reintroduced as **Track Corner Acid Trap** on the **DRC** dialog. This check is much faster than the full Acid Trap DRC but only checks track corners, hence, it isn't as comprehensive but is less complicated to set up.



Copper Check Rule Changes

Important changes have been made to the existing **Copper Check Rule** dialog under **DFM/DFT** in the **Technology**; the algorithm for the way checks are performed has been significantly updated to add enhanced accuracy.

The check within this dialog that provided both Sliver and Antenna checks has been changed so that both can be defined separately in different rules dialogs; the Sliver check under the **Copper Check Rules** dialog now uses a different, updated, algorithm as discussed above. See below for the new Copper Antennae Rule.



The copper sliver check is more comprehensive and will detect any long thin piece of copper (a copper sliver) or anywhere the copper thins down too much (a copper bottleneck). This is defined using the **Max Sliver Width** value is will be an absolute sliver thickness or as part of the min sliver perimeter value defined.

Dialog Parameters

The previous **Min Sliver Length** value has been replaced by the **Min Sliver Perimeter** value which reflects the updated algorithm.

The **Min Copper Width** rule and check hasn't changed.

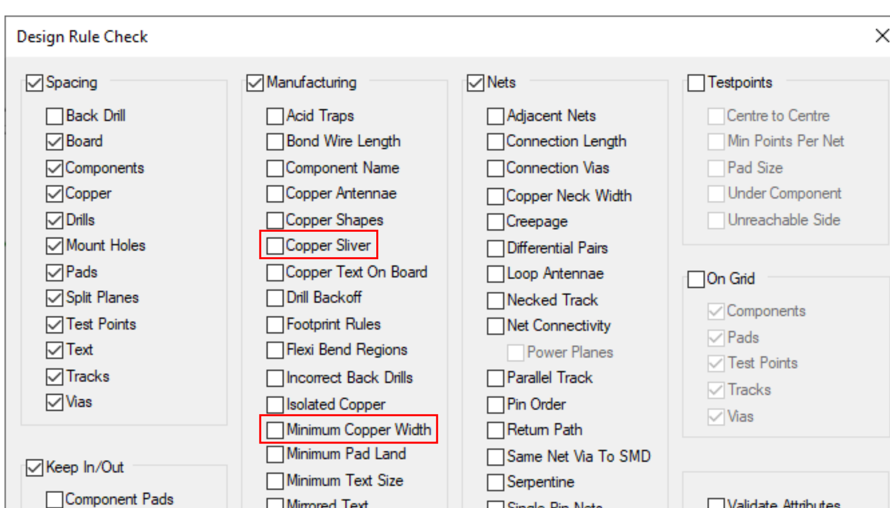
The **Min Copper Angle** is the minimum required angle between two copper segments to avoid sharp angles being created, which can result in **copper slivers**. This is used by the **Minimum Copper Width** check in DRC.

The **Max Sliver Width** is the maximum width required for a copper section to be included as a sliver by the **Copper Sliver** check in DRC.

The **Min Sliver Perimeter** is the minimum perimeter distance from the start to the end of a copper sliver and is checked using the **Copper Sliver** check in DRC.

New DRC Check

New checks have been introduced for **Minimum Copper Width** and **Copper Sliver** in DRC.

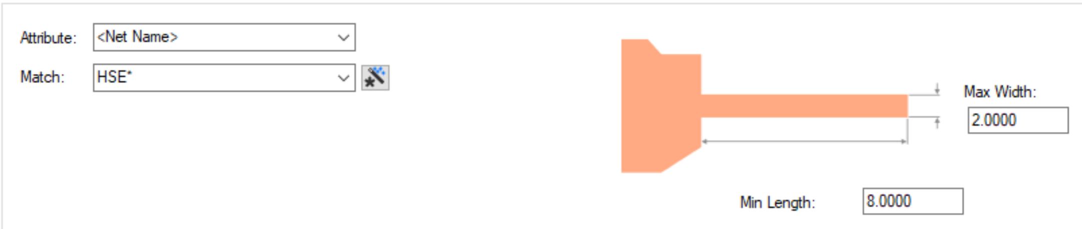


New Copper Antennae Rule

There is a new **Copper Antennae Rule** under **DFM/DFT**. This rule was previously defined under the **Copper Check Rule** and is now a completely separate check, with its own rules.

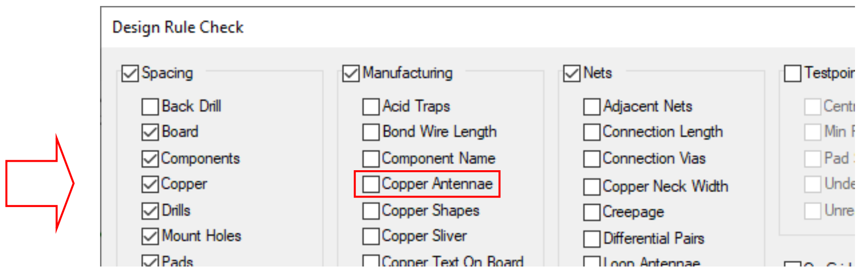
A copper antenna is a long, thin, dangling section of copper that can act as an antennae for high speed designs.

The Copper Antennae Rules dialog is used to define the maximum width and minimum length of copper which would be considered an antenna.



New DRC Check

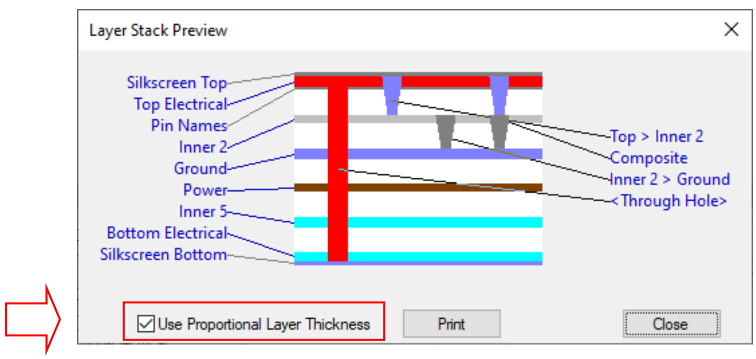
A new check has been added to the **DRC** dialog under the **Manufacturing** section named **Copper Antennae**.



Layer Stack – Use Layer Thickness

A new feature has been added to allow the display of the **Layer Stack** to **Use Proportional Layer Thicknesses**.

There is a new check box to do this on the **View Layers** window available from the **Technology Layers** dialog.



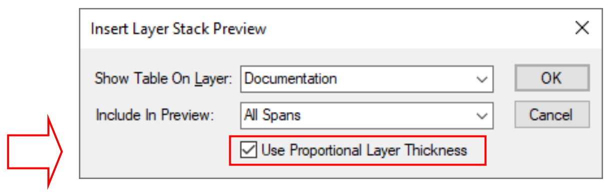
This switch and feature is also available in the **Layer Spans** dialog, and in the **BackDrill** dialog.

Select the **Use Proportional Layer Thicknesses** check box if you wish to represent the preview using a proportional representation of the actual layer thicknesses if defined in the **Technology - Layers** dialog along with thicknesses defined in the **Technology - Materials** dialog.

When this check box is unchecked, the layer stack preview will use a fixed size for non-electrical, electrical and construction layers (as it did previously)..

Insert Layer Stack

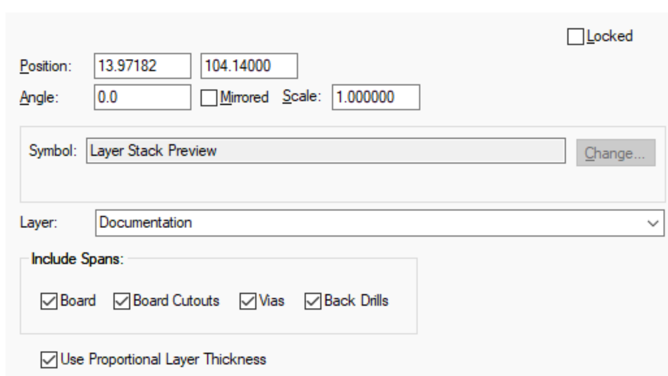
This new check box has also been added to the **Insert Layer Stack** dialog,



Note: When selecting the Layer Span for use in this dialog (**Include In Preview**), there is a limited choice. If you wish to expand the selection to other Spans, add the Stack to the design and then use **Properties** dialog of the selected symbol to select the Span required.

Properties of an inserted Layer Stack

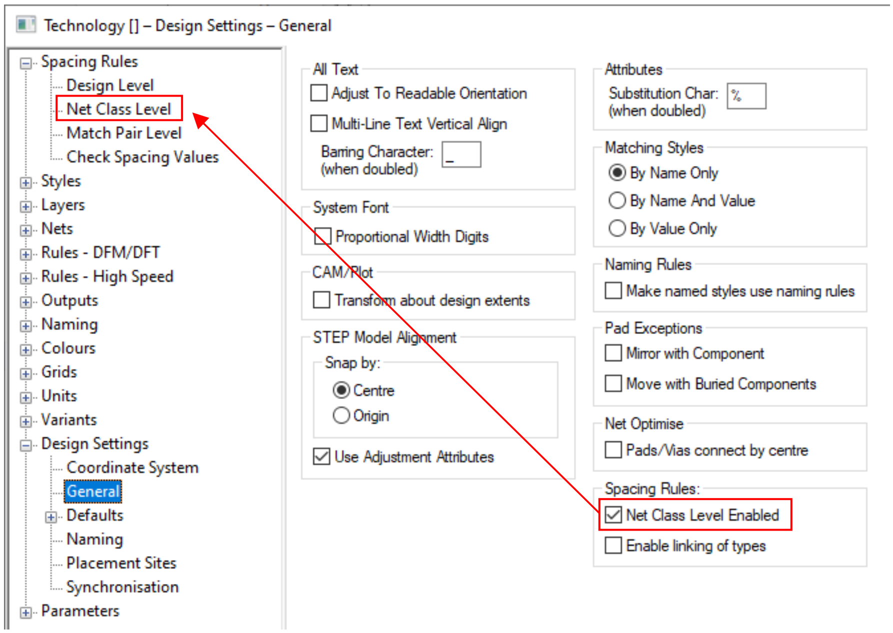
It is also available in **Properties** of an inserted **Layer Stack Preview**.



The **Include Spans** drop down box has been replaced with check boxes enabling you to select the Layer Span required in the Preview in the design. If you wish to have more complexity, then you should use the **Report Maker** option to create a more specific report that is then added to your design using the **Insert User Reports** option.

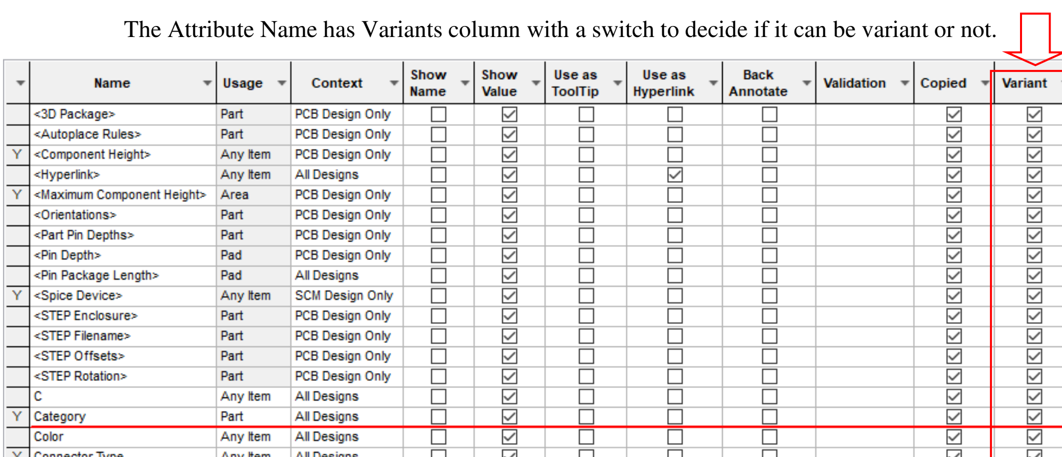
Ability to Switch Off Net Class Level Spacings

You can now disable **Net Class Level Spacing** rules if you do not use Net Classes in your design and this dialog is not relevant. Use the **Design Settings - General - Spacing Rules** dialog and **Net Class Level Enabled**. Unchecking this option will remove any Net Class Spacings and not show the **Net Class Level** page on the Technology dialog.

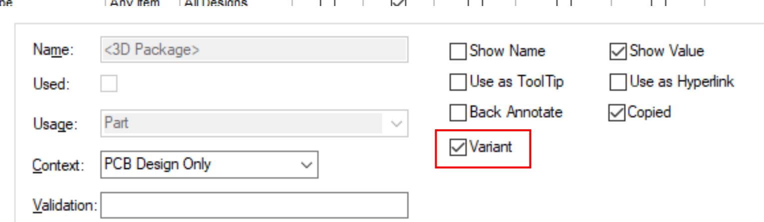


Disable Variant Attributes

The Attribute Name has Variants column with a switch to decide if it can be variant or not.



	Name	Usage	Context	Show Name	Show Value	Use as ToolTip	Use as Hyperlink	Back Annotate	Validation	Copied	Variant
	<3D Package>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<Autoplace Rules>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
Y	<Component Height>	Any Item	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<Hyperlink>	Any Item	All Designs	☐	☑	☐	☑	☐		☑	☑
Y	<Maximum Component Height>	Area	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<Orientations>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<Part Pin Depths>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<Pin Depth>	Pad	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<Pin Package Length>	Pad	All Designs	☐	☑	☐	☐	☐		☑	☑
Y	<Spice Device>	Any Item	SCM Design Only	☐	☑	☐	☐	☐		☑	☑
	<STEP Enclosure>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<STEP Filename>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<STEP Offsets>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	<STEP Rotation>	Part	PCB Design Only	☐	☑	☐	☐	☐		☑	☑
	C	Any Item	All Designs	☐	☑	☐	☐	☐		☑	☑
Y	Category	Part	All Designs	☐	☑	☐	☐	☐		☑	☑
	Color	Any Item	All Designs	☐	☑	☐	☐	☐		☑	☑
Y	Connector Type	Any Item	All Designs	☐	☑	☐	☐	☐		☑	☑

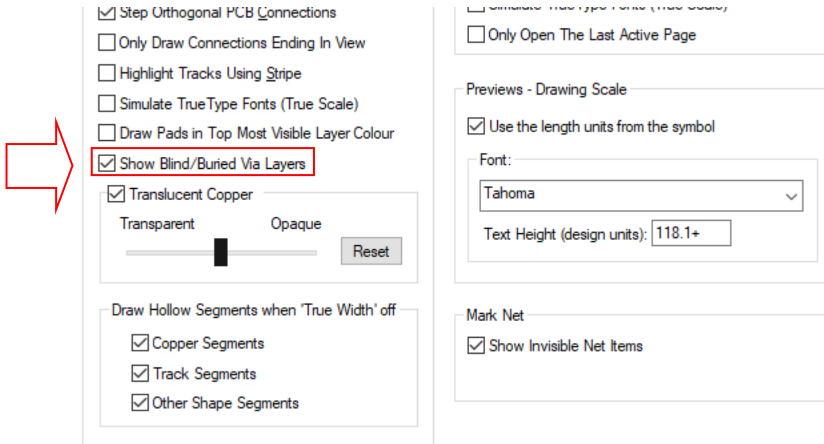


With this switch unchecked, attributes cannot be Variant specific. It does not apply if the attribute has already been used with a different value to that of other variants.

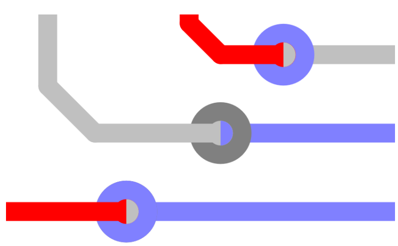
Visual Indicators of Blind / Buried Via Layer Spans in the Design

The ability to display the ‘outer’ most track colours within a blind and buried via stack has been added. This will show the layers that they connect for easier identification. This also includes Micro-vias and Composite vias.

This option is off by default and can be enabled through the **Options** dialog, and **Display** page using the **Show Blind/Buried Via Layers** switch.



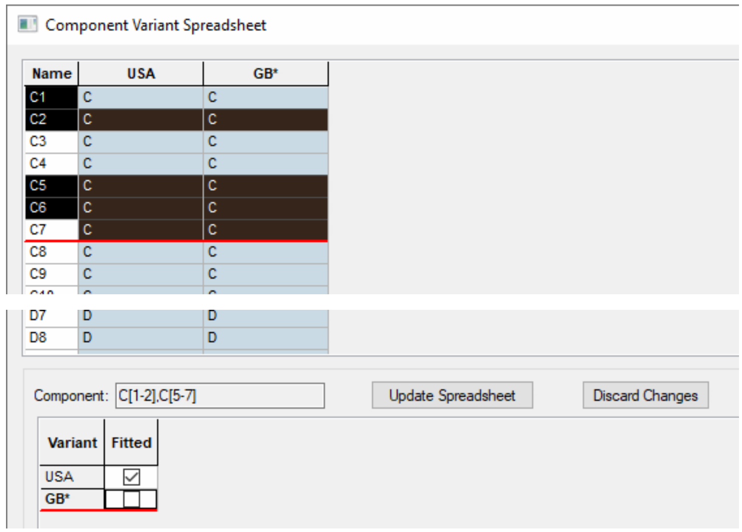
When set, it draws half of the via in each of the span electrical layer colours. The ‘highest’ electrical layer colour is shown in the left side of the via, and the ‘lowest’ layer electrical colour on the right side of the via.



Component Variants

Multi-selection Fit / Unfit

You can now select multiple components with simplified variants in the **Component Variants Spreadsheet** located on the **Edit** menu and then **Fit** or **Unfit** them by variant.



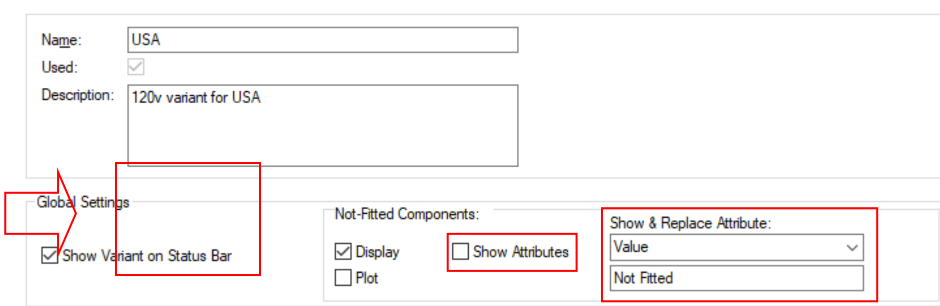
This feature was back-fitted to V12.5.

Turn off display of Attributes on Unfitted Components

A new option (**Show Attributes**) has been added to the **Technology** dialog in the **Variant Manager**.

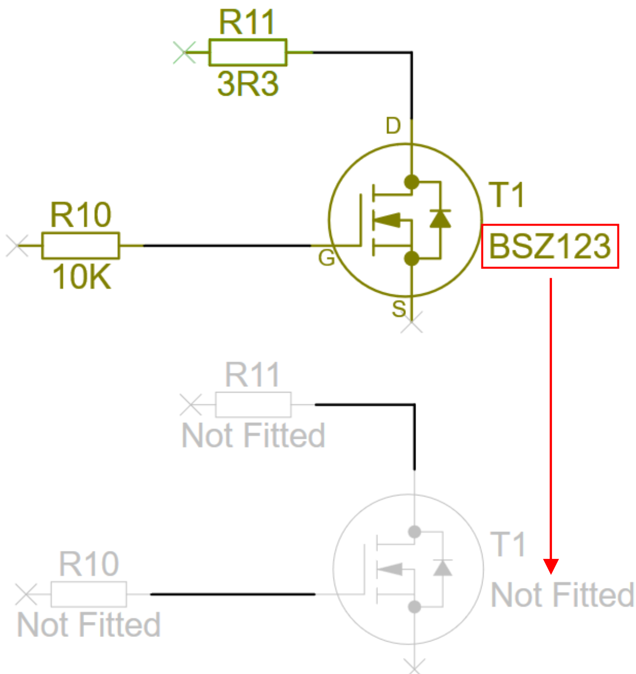
This option would normally be checked to Show Attributes.

Uncheck this option to not show attributes on an unfitted component. The Attribute box and text box are then available. The only attributes which will be shown are the Component Name and Pin Names.



Once unchecked, You can also optionally show a selected attribute and replace the value with the fixed text specified. For example you could specify that the Value attribute (*Value* in our example below) could be substituted with the words *Not Fitted* in the design when a variant is used.

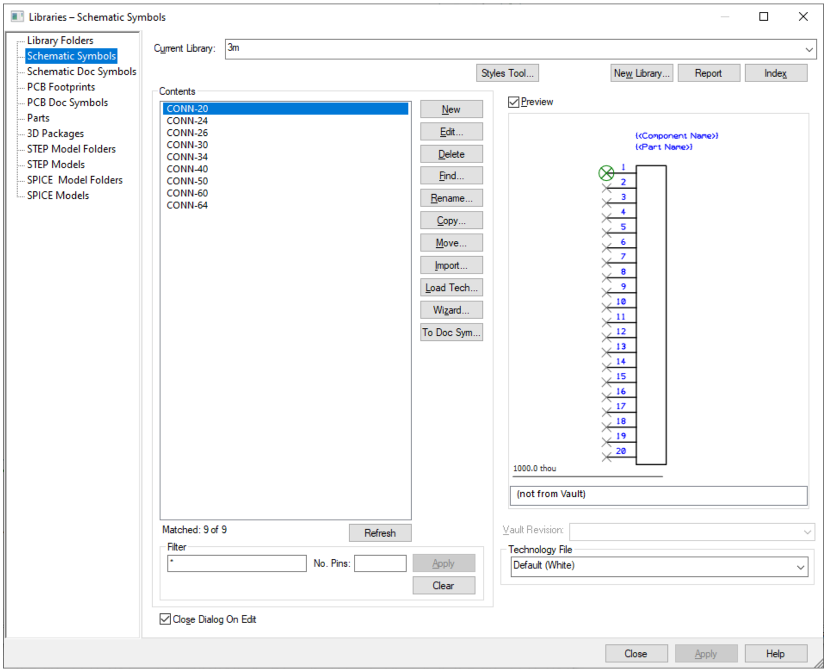
The design below illustrates this:



Library Manager Changes

Updated Library Manager Dialog

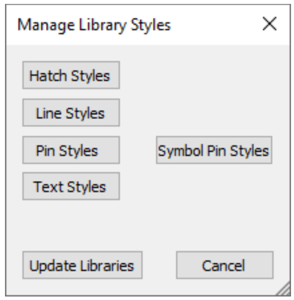
The **Library Manager** dialog has been rearranged to better support additional functionality for **SPICE Models**.



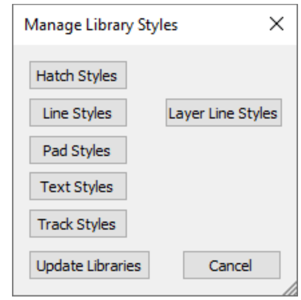
New Styles Tool Option

In the **Library Manager**, a **Styles Tool** button has been added to the **Schematic Symbols**, **Schematic Doc Shapes**, **PCB Footprints** and **PCB Doc Symbols** pages. This option enables you to manage **styles** and **names** within selected library items. You may find that library items created from external sources or different people in your organisation have alternative names, this tool allows you to standardise and rationalise styles.

Schematic Symbols & Doc Symbols

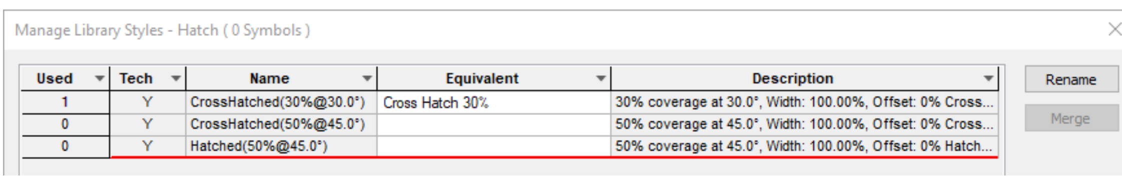


PCB Footprints & Doc Symbols



Each of the dialogs (Hatch, Line, Pin, Pad, Text and Track) has the same layout and perform the same function. They also handle name rules and templates.

The **Update Libraries** button makes changes made permanent to any styles and closes the dialog. If you use **Cancel**, this will remove any changes made to styles in the current editing session.



The number of symbols using each style is shown in the **Used** column.

If the style is in the Technology file, the **Tech** column shows 'Y'; if not, it shows 'N'.

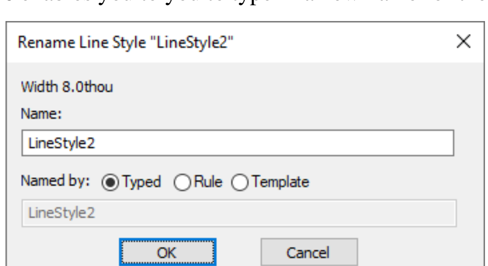
You can use the **Selected** or **All** buttons to report **where** styles are **used**.

Using the **Selected** button will report, for each style, the number of symbols using it. If the style is used, the symbols will be listed based on their file path. Likewise, using the **All** button will report the same information for every style that is present in the dialog.

Where styles match, the **Equivalent** field will display the name of the matching style.

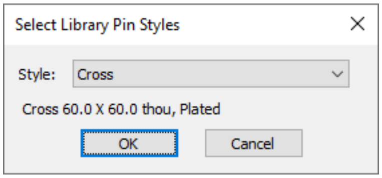
If you have only one style selected with equivalent styles, clicking **Merge** will combine all of the equivalent styles with the one you have selected. Otherwise, the **Merge Styles** dialog box will appear if there are several styles selected, asking you to decide which style you wish to preserve. The selected style will replace the other styles.

Rename enables you to you to type in a new name for the selected style.

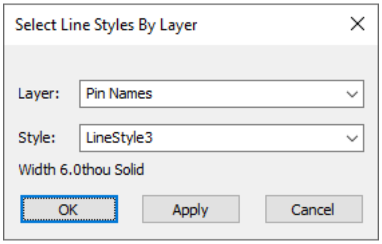


You can also named your style based on a rule or template. If there is already a style with that name and they are equivalent then the two styles will be merged. If they are not equivalent you will be prompted to see if you want to merge them. In case the style is in technology file, a popup warning will appear, informing you that the style name will not be changed in the technology file.

Using the **Symbol Pin Styles** allows you replace the pin style for all the selected Schematic symbols in the libraries to the one chosen.

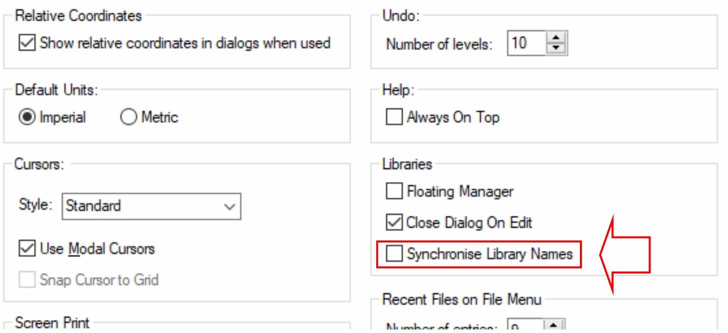


Using the **Layer Line Styles** allows you to select a line style to be used for all shapes on a particular layer for all selected footprints. You may wish to use a named style, for example Component Outline, on all Top Silkscreen layers on your footprints. This dialog enables you to achieve this.



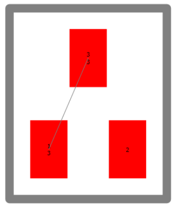
Synchronise Library Names Across All Library Tabs

A new option has been added in **Options, General** under the **Libraries** section, named **Synchronise Library Names**. With this enabled, if you change one **Library name**, the Parts page for example, the same name will be used in the corresponding 'other' Schematic Symbol and PCB Footprint pages. It will only be changed if the corresponding library name exists.



Connections between pads in Footprint

Within the **Footprint Editor**, **Pads** on **same net** now have connection guides drawn between them.



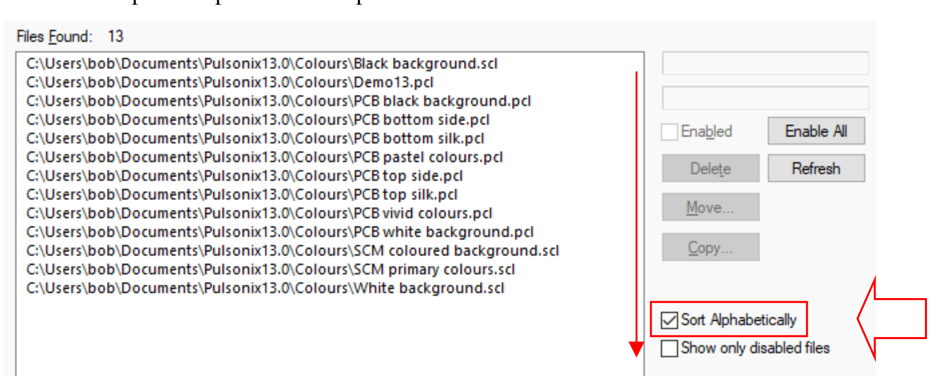
The colour for the connection guide can be changed in the **Colours** dialog under the **Others** page using **Connections**:

Name	Displayed	Selectable	True Width	Colour
Background				
Branch Point Name	☑			
Branch Point Origin	☑	☑		
Connections	☑	☑		
Coordinate Origin	☑	☑		
Net Name in Tracks	☑			
Pad Name Background	☑			
Pad Name Text	☑			
Relative Coords Origin	☑	☑		
Spacing Shapes	☐			
Symbol Origin	☑	☑		
Force Everything	☐	☐	☐	

Folders

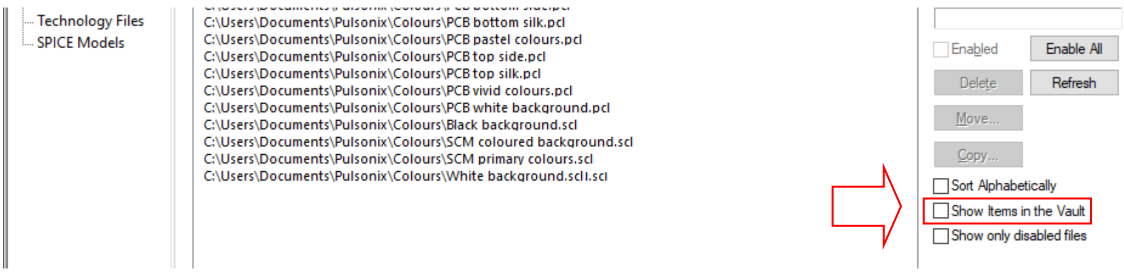
Sort Files Found Alphabetically

On each of the folder selection pages within folder, there is now a **Sort Alphabetically** button to enable you to sort the **Files Found** list. It will sort the files in the folder in alphabetical order regardless of the folder they appear in. With it left unchecked, the files will be sorted in folder order then file type then name. This switch only affects the files found list in the Folders list and has no bearing on any other library items found, such as in the Parts Editor list in the Library Manager or the Insert Component option for example.



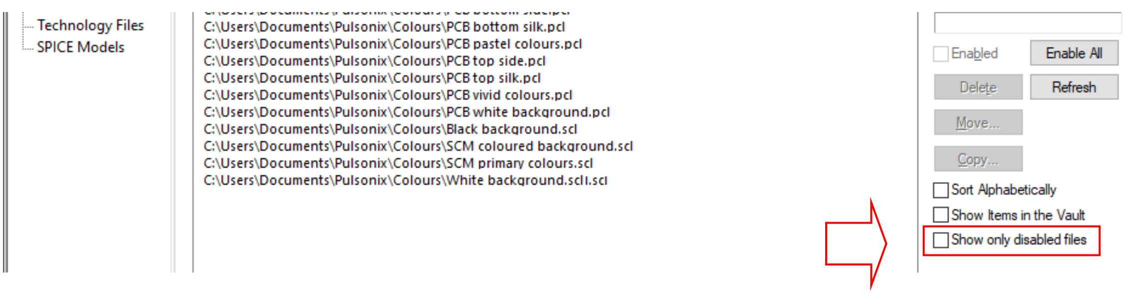
Show Items in the Vault

When the Vault is in use, the new **Show item in the Vault** switch can be used to display individual items found only in the Vault. Items found outside of the Vault will not be displayed. With this option left unchecked, items found in both the Vault and outside of it will be displayed.



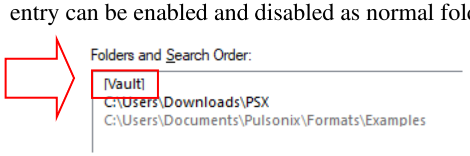
Only Show Disabled Files

A new check box option, **Show only disabled files,** has been added to folder pages that show lists of files, such as the Libraries page. With this selected, it will only list any disabled files. This gives you the ability to quickly view them in the list and re-enable them.



Vault listed as a Search 'folder'

In the **Folders** dialog, You can list **Profile Files** and **Technology Files** from the **Vault**. The Vault entry can be enabled and disabled as normal folders.



Performance Enhancements

DRC option

DRC speed has been improved with performance enhancements, both, with and without multi-threading enabled.

Multi-threading in DRC has been improved to make better use of system resources. Large designs performing multiple checks can be up to 70% faster.

Component name checks have also been significantly sped up. On very large designs, run times have been reduced by up to 85% (even with multi-threading disabled).

Optimise Nets

Net Connectivity DRC

Following the implementation of new multi-threading techniques, the **Optimise Nets** option and the **Net Connectivity DRC check** have seen performance enhancements when multi-threading is enabled.

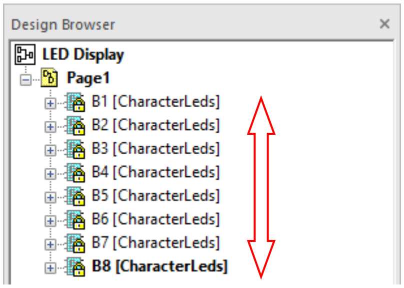
An additional improvement has been made more specifically where a higher thread count (8 threads or more) will no longer cause Pulsonix to run slower than a medium thread count (4 – 8 threads), due to the new multi-threading technique running more efficiently with system resources.

Additionally, the time it takes to complete **Optimise Nets** or **Net Connectivity** checks has been significantly reduced when using a higher thread count. On large designs, run times of up to a 60% reduction have been seen.

Design Browser

Ability to Drag Schematic Pages

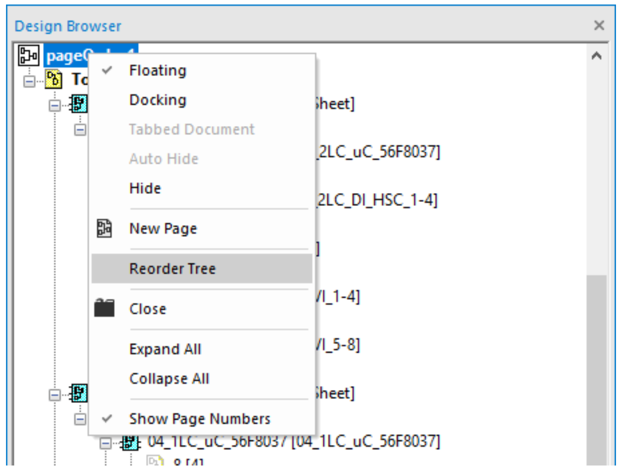
If you are using multiple pages or hierarchy in your Schematic design, you can now drag and drop a design within the **Design Browser** list to change the order.



Re-order Design Browser Tree on context menu

After dragging blocks or pages in design browser tree, the page numbers may be out of (numerical) order, you can now right click and select **Reorder Tree** from the context menu.

Note: This option is only available from the top design level.

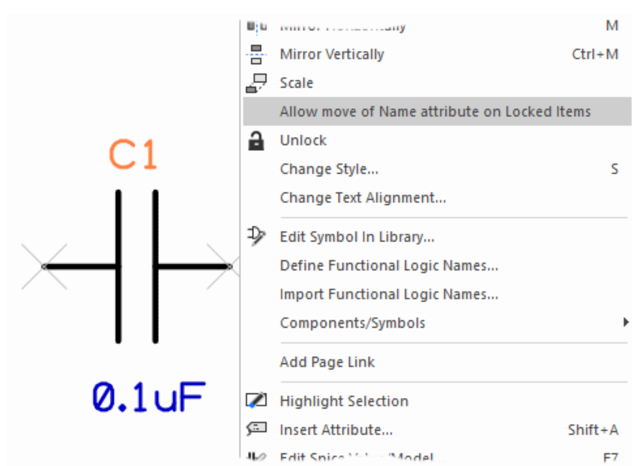


Move Name Attribute option on a Locked Component

When a Component Name is locked (locked name or locked component), when it is selected, there is a new option available on the context menu - **Allow move of name attribute on Locked items**. This is a toggle item that overrides the lock status.

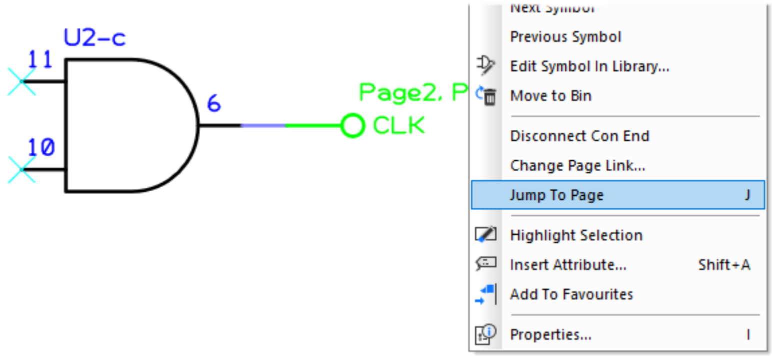
If you want to disable the option, right clicking again on the selected component or name allows you to select the option, **Don't allow move of Name attribute on Locked items**.

These options are also available when multiple components are selected.

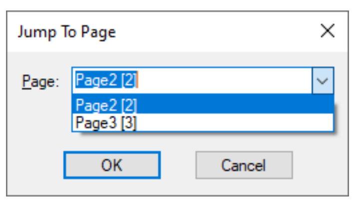


Net Pages Attribute – Jump To Page

The option to **Jump To Page** appears in the context menu when you right click on a **net pages** attribute.



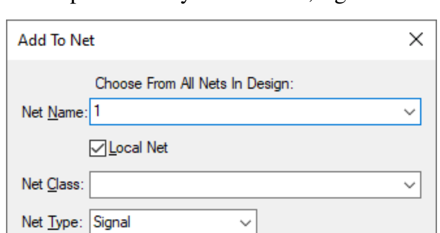
When used, if there is only one other page on the attribute you will be redirected to that page. If there are multiple pages to choose from, a dialog will pop up with a drop down list of pages and you will be redirected to the page you select.



Ability to Add a Pin to a Net in the SCM Symbol Editor

The ability to **Add to Net** has been added for a selected pin within the Schematic **Symbol editor**. With a net assigned to a pin, it means that you can now also assign a Net Attribute to the pin (see below).

For a selected pin in the Symbol Editor, right click and select **Add To Net** from the context menu:



Ability to Predefine a Net Attribute on a Pin

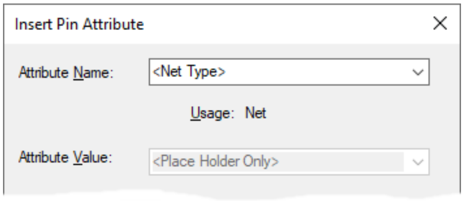
The ability to assign a **Net Attribute** assigned to a **Schematic Symbol** pin has been added. In order for this feature to be available, the pin must already on a net (see above, **Add To Net**).

By assigning a Net Attribute to the pin, it means that when the Symbol is used on a Part in the design, a connection added from that pin will display and inherit the net properties. It also means that the target pin inherits any net attributes defined, for example, the *Net_Driver*.

Adding a Net Attribute to a Pin

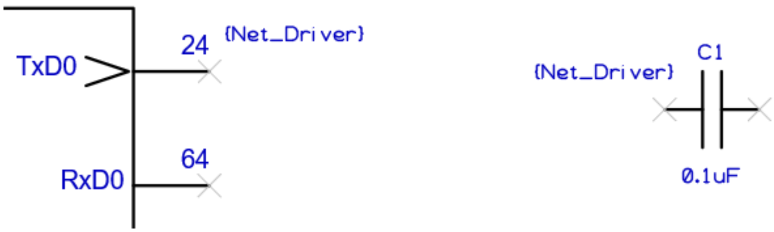
In the **Schematic Symbol Editor**, once the pin has added to a net using **Add To Net** (see above), you can now add the **Net Attribute** to a Pin. You can use either **Properties** or **Insert Attribute** from the context menu on a selected pin or from the **Insert** menu.

The **Insert Pin Attribute** dialog now allows you to select a Net Attribute type. The usage will be shown as **Net**

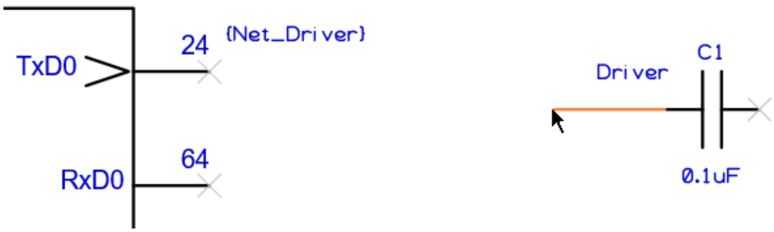


Using Net Attributes on a Pin in the Design

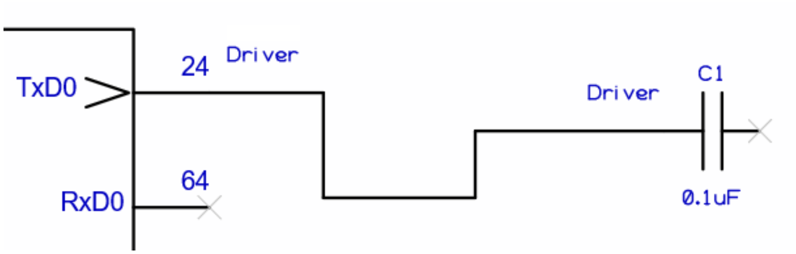
When applied to a pair of pins (as below), you see it like this in the design:



When a connection is then drawn off that pin, the Net_Driver attribute name, *Driver* in our example, is displayed in the design:



Completing the connection will display the fulfilled Net_Driver attribute name on the target pin also as it has now inherited the net attribute name.

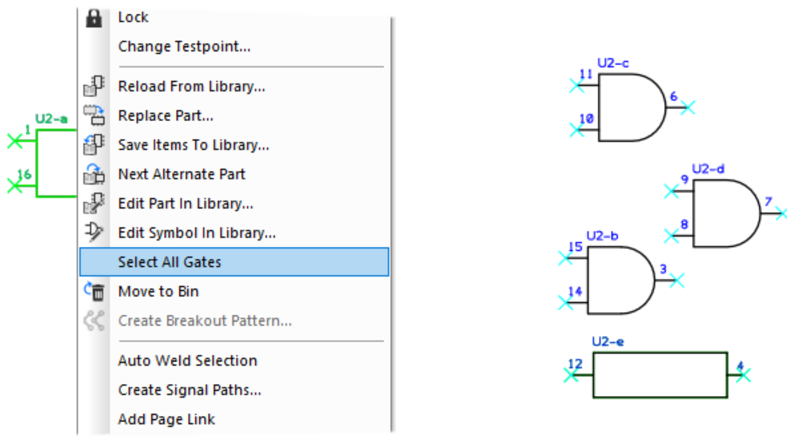


Change Description Attribute in Parts Library using Import CSV

The **Description Attribute** can now be changed using the **CSV Import** option from within the **Library Manager**, **Parts** and **Edit Attributes** dialog.

New Select All Gates feature in Schematics

In Schematics, an option to **Select all Gates** appears on the context menu when you select a single gate or Connector in a Schematic design. This option enables you to select all the gates associated with that Component in the design. This option works across all sheets in the design.

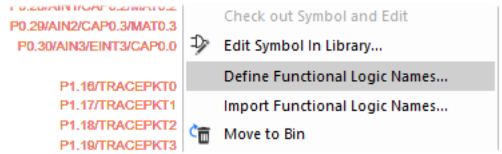


Once selected, all the gates will be selected in the design:

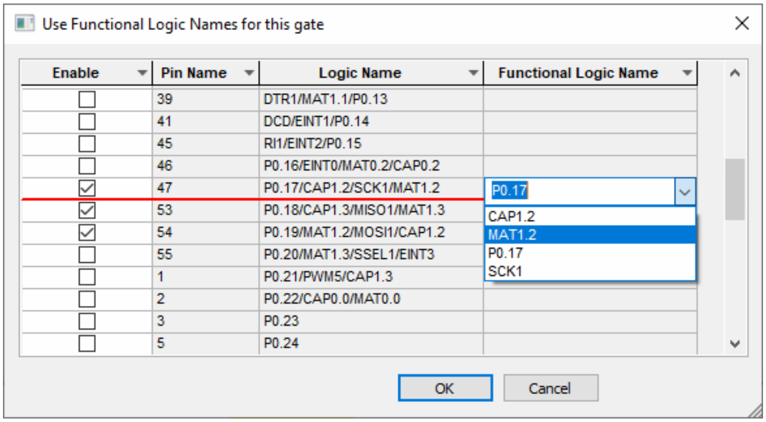
Select Functional Logic Name of a Part Pin

In a Schematic design, a new option has been added called **Define Functional Logic Names**. This is available on the context menu when a single Gate or Gates are selected. If selecting multiple Gates, they must be part of the same Component.

This option enables you to create the Part Logic Names as defined on the manufacturer’s datasheet but to then choose the actual Logic Name in the design based on the pin function.

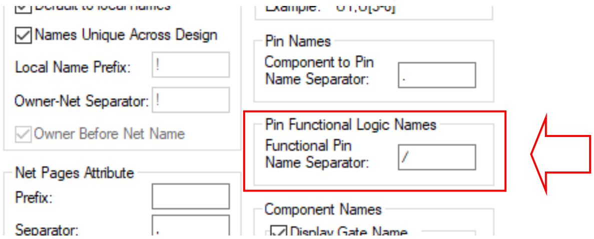


On selection, a dialog displays the gate's pins and their Logic Name.



When the check box **Enable** is selected, you can either select one of the available **Functional Logic Names** that appear in the combo box or type your own.

A drop-down list will be available if multiple logic names are found based on the your defined **Functional Pin Name Separator** in the **Design Settings** – **Naming** dialog under **Pin Functional Logic Names**..



Once defined, the logic name is then updated on the Gate pin in the Design.

Local Editing of Logic Names

Using the new **Define Functional Logic Names** option, you can also edit a logic name manually to provide a name of your choosing.

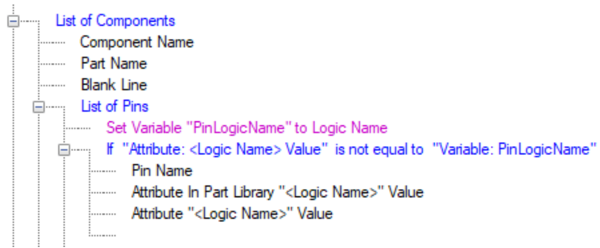
Resetting Logic Names

If you wish to reset the design Logic Name values of the gate to those defined for the Part in the Parts Library, use the **Reload From Library** option.

Reporting Modified Logic Names Using Report Maker

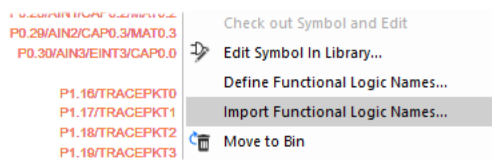
Within the **Report Maker**, there are two commands under **List of Pins** that enable you to report the **Attribute in a Parts Library** (the Logic Name) and an **Attribute** equalling the **<Logic Name>**.

By using a **variable** within the List of Pins to set the **Part Logic Name** of the **Parts Library** definition, you can test to see if it is different to that of the **Design gate Logic Name**.



CSV Import of Logic Names

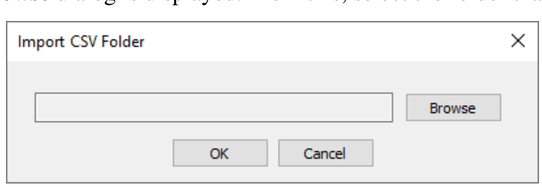
In a Schematic design, a new option has been added called **Import Functional Logic Names**. This is available on the context menu when a Gate is selected. This enables you to import Functional Logic Names from your FPGA development system for example, to create and map actual logic names required.



If the import file has pins mapped for more than one gate, but the other gates are not selected, then only pins for the selected gate will be imported. If the import file represents pins for the whole component over multiple gates, then use the Select All Gates option first from the context menu.

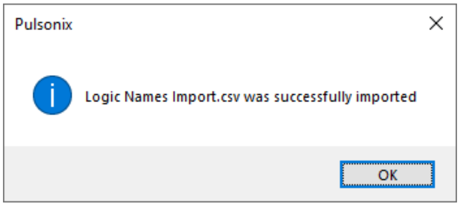
With a Gate selected in the Schematic design, right click and select **Import Functional Logic Names** from the context menu.

The **Browse** dialog is displayed. From this, select the folder that contains the CSV file.



With the folder selected, the **Open** dialog is displayed. From this, select the CSV file to import.

It will process the CSV file and if successful, will report with a confirmation dialog. A text report will also be produced if there are any errors.



The CSV File Format

The format for the .csv file has just two fields, separated with a comma.

The first line is the header and is ignored on import.

The second and subsequent lines are the Pin Number and new Functional Logic Name (separated with a comma). If a space is used after the comma, this is ignored. The Pin Number can be numerical or alphanumeric.

```
Pin_Number, Functional_Logic_Name
19, TxD0
21, RxD0
47, MAT1.2
53, MAT1.3
B12, CAP1.3
Y11, RST
```

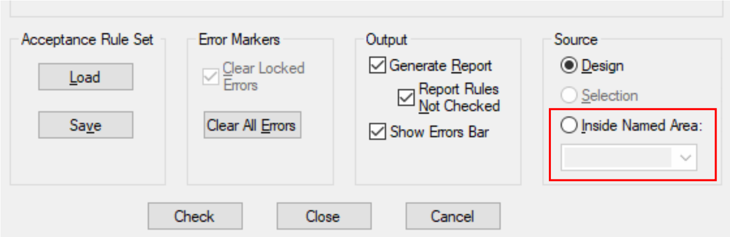
New Electrical Rules Checks (ERC) in Schematics

New checks have been added to the Schematic ERC option:

Perform ERC Inside a Named Area

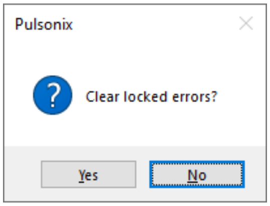
You can now opt to perform checking **Inside Named Area** in the **Electrical Rules Check** dialog under the **Source** group box.

The radio button is available only if there is a **named area** defined in the design. If there are multiple named areas, the drop-down list will contain all the available named areas in the design.



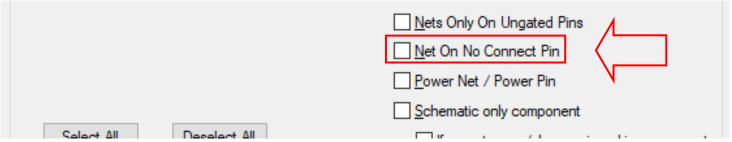
Delete Locked Errors

On running ERC, if locked errors are present, the option will now prompt to **Clear locked errors?** The option is defaulted to No, (to not clear locked errors).



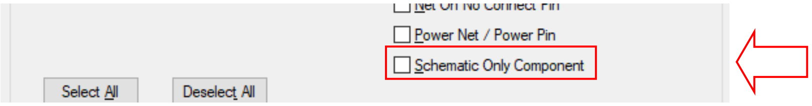
Net on No Connect Pin

A new ERC check has been added - **Net on No Connect Pin**. This checks for any No-Connect pin types that are connected to a net.



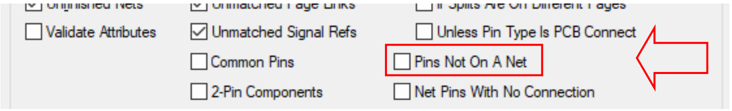
Report Schematic-only Components

A new ERC check has been added - **Schematic only component**. This checks for Schematic-only Components in the design. That is, Parts with no footprint assigned and thus will not translate to PCB.



Pins Not On A Net check excludes Mounting Hole or Ancillary Pad Pin Types

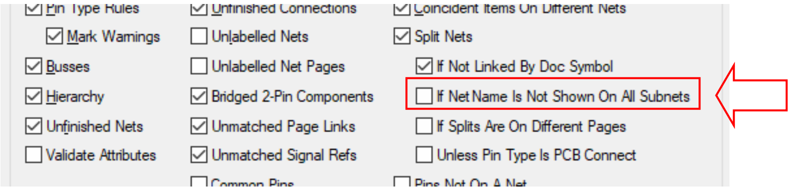
The existing check, **Pins Not On A Net** has been extended so that if a pin is marked with a **Pin Type** of **Mounting Hole** or **Ancillary Pad,** they are excluded from checking, regardless of whether they are on a net or not.



Split net – If net name is not shown on all sub-nets

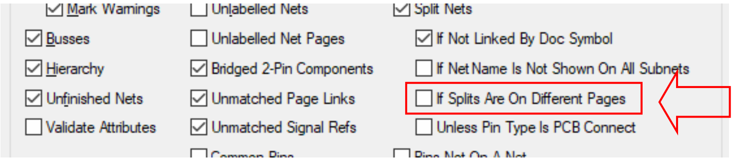
There is a new Split net option available in ERC - **If net name is not shown on all sub-nets**. Select this check to ignore the split net error if all the sub-nets have a net name attribute position.

For example, you may have a named net that is split into two or more sub-nets and uses a Doc Symbol to visibly 'label' the net. With the presence of a net name attribute position (i.e. Display Net Name on all sub-nets) the error can now be suppressed.



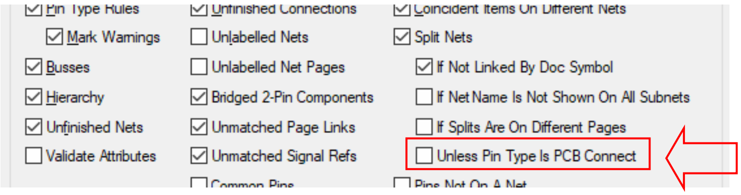
Split net – If Split Nets are on a Different Page

There is a new Split net option available in ERC - **if Splits are on different pages**. Selecting this check ignores the split net error if it is on the same page.



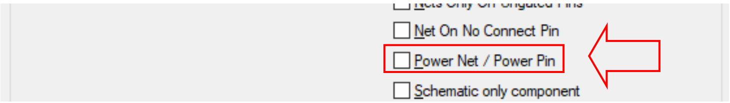
Split net – Unless Pin Type is No Connect

There is a new Split net option available in ERC - **Unless Pin Type is PCB Connect**. The split net error is ignored if the **Ungated Pin** is a Pin Type of **PCB No Connect**.



Power Nets on Non-Power Pins

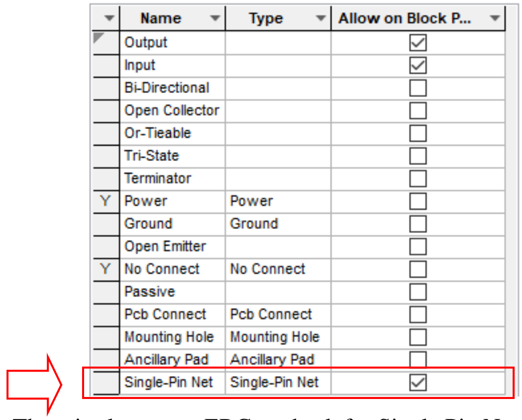
A new ERC check has been added - **Power Net/Power Pin**. This checks if non-power nets are connected to power pins.



Technology - New Pin Type – Single Pin Net

A new **Pin Type** has been added to the **Technology Pin Type** dialog, called– **Single Pin Net**

This pin type can be added to a Component pin to specifically define its use and override it as a single pin net.



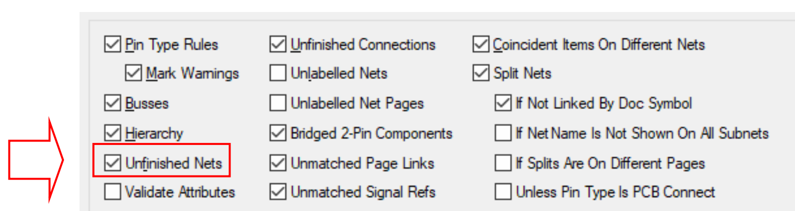
	Name	Type	Allow on Block P...
	Output		☑
	Input		☑
	Bi-Directional		☐
	Open Collector		☐
	Or-Tieable		☐
	Tri-State		☐
	Terminator		☐
Y	Power	Power	☐
	Ground	Ground	☐
	Open Emitter		☐
Y	No Connect	No Connect	☐
	Passive		☐
	Pcb Connect	Pcb Connect	☐
	Mounting Hole	Mounting Hole	☐
	Ancillary Pad	Ancillary Pad	☐
	Single-Pin Net	Single-Pin Net	☑

There is also a new ERC to check for Single Pin Nets (see below).

Single Pin Net Check

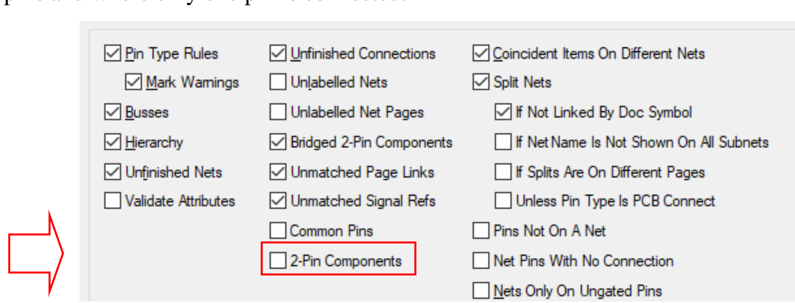
An extension to the **Unfinished Nets** check in **ERC** to check for the **Single Pin Net** attribute. This check enables you to distinguish between pins specifically marked with a Single-Pin Net attribute and those accidentally left as single pin nets.

Note, there is a new Technology Pin Type of Single-Pin Net to accommodate this check.



Check for 2-Pin Components

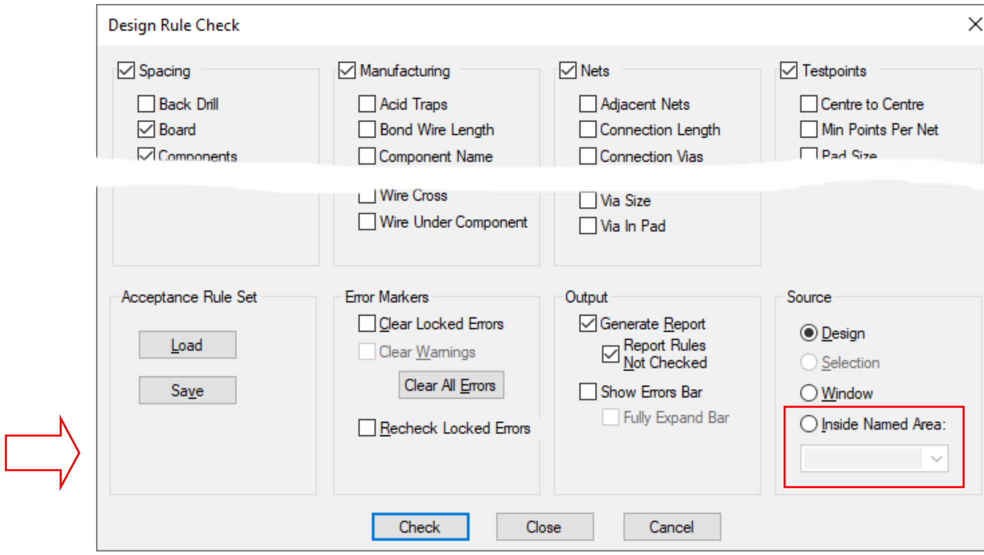
A new check is available in **ERC** to check **2-Pin Components**. This checks for components with two pins and where only one pin is connected.



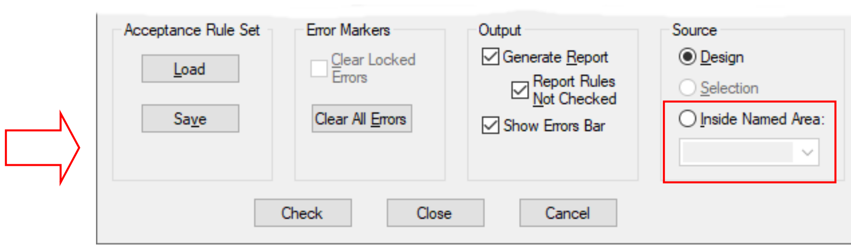
DRC - Inside a Named Area

For both **Electrical Rules Checking** in Schematics and **Design Rules Checking** in PCB, you can now check **Inside Named Area**.

The radio button is only available if there is a named area defined in the design. If there are multiple named areas, there will be a drop-down list with all the available named areas in the design.

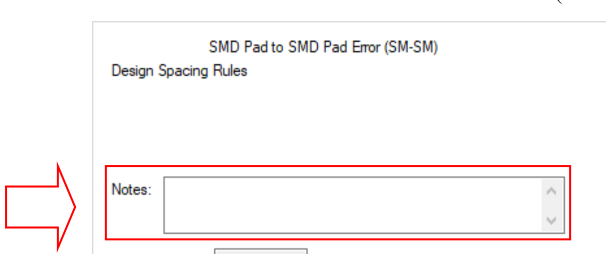


And ERC dialog:



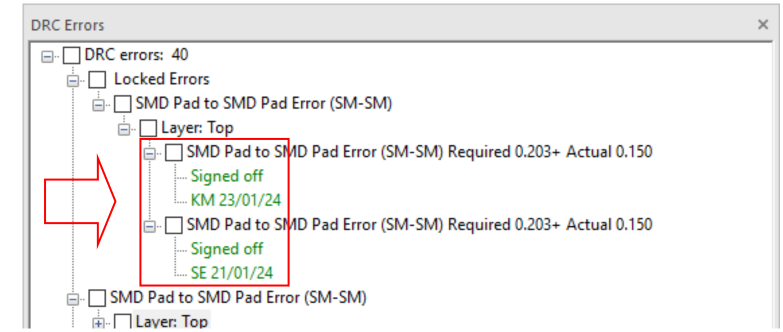
Multi-line Notes available in Properties dialog for DRC / ERC Error Markers

Multi-line Notes are available in Properties for DRC/ERC error markers. These will then appear in the **DRC/ERC Error bar** under the **Locked Error** list (see below).



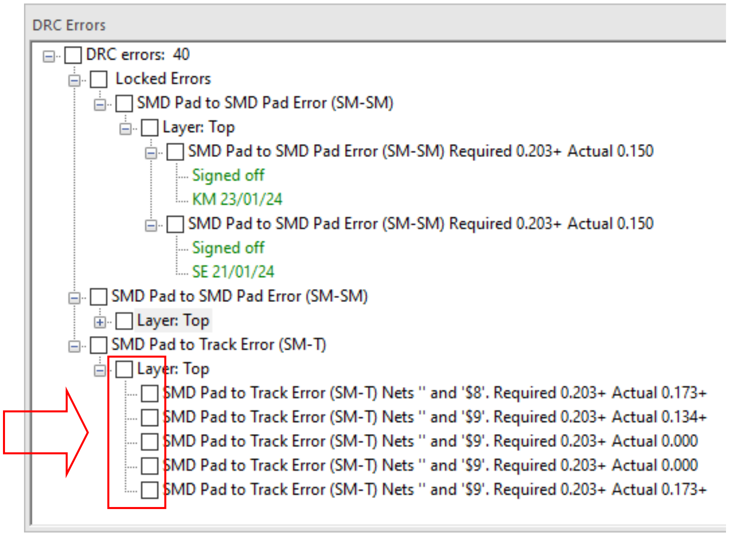
Notes visible for Locked Errors in DRC / ERC Errors Bar

Notes and multi-line Notes for **locked errors** are now shown in Green under the error in the **DRC/ERC Errors Bar**.



Multi-select Errors in DRC / ERC Errors Bar

DRC and ERC Error Markers are now displayed with a check box next to them, this indicates that they can be selected. You can select multiple errors, this allows you to perform additional operations on them, such as Locking, Unlocking or Deleting.



IDX Collaboration Interface

A new option has been added to the **3D Design** menu, called **IDX Collaboration**. This option could be used as an alternative to the STEP format, used for integration with mechanical CAD packages.

What is IDX?

IDX (Incremental Design Exchange) collaboration refers to a process and technology standard used to facilitate efficient and accurate file transfer between PCB CAD (ECAD) and Mechanical CAD (MCAD) systems.

IDX enables the seamless exchange of design data between ECAD and MCAD software, products such as SolidWorks. One of the core benefits of IDX is the ability to exchange incremental design changes, thus eliminating large design transfers each time a change is made. Changes can be tracked and reviewed with each change accepted or rejected.

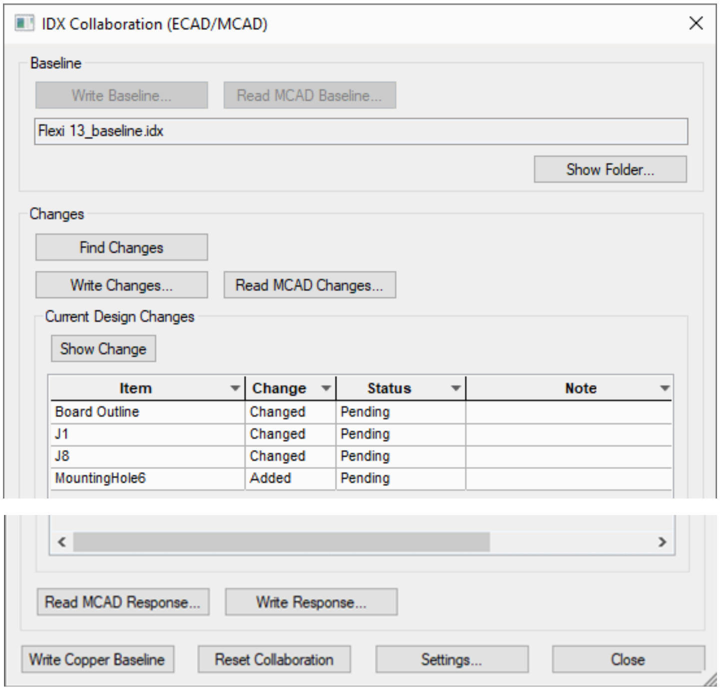
Why use it

IDX provides an alternative format to that of STEP or DXF for integration with Mechanical CAD (MCAD) packages. Its main advantage is one of the transfer speed to the MCAD system and subsequent changes, which are incremental. However, not all MCAD packages support this format, so the other file types supported would need to be used in these circumstances.

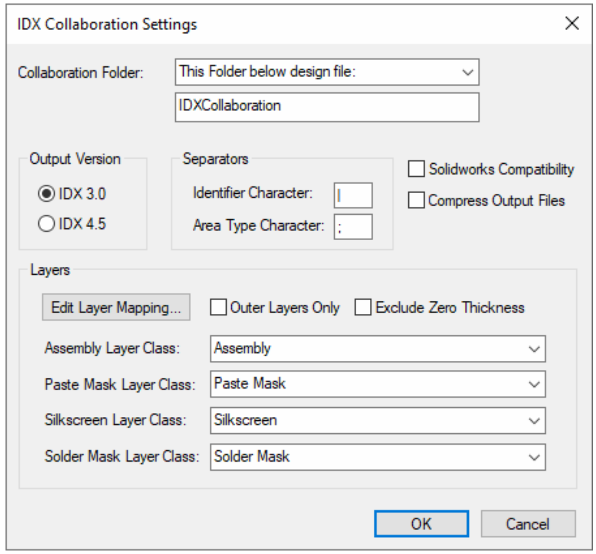
Using the dialog

Available from the **3D Design** menu, **IDX Collaboration** option.

On selection of this option, the main IDX Collaboration dialog is displayed. This enables you to control the writing and reading of IDX files for both baseline and incremental files. IDX also has control over changes; whether they are accepted or rejected. For example, an IDX file written from Pulsonix may be rejected on a component clash on the case or enclosure in the MCAD system.



The **Settings** button displays a dialog that enables you to setup the different options available used when running the IDX Collaboration feature:



Workflow

Writing an IDX baseline will happen once at the very start of the collaboration so that both ECAD and MCAD are synchronised. Afterwards, both the workflows described above (ECAD to MCAD, and MCAD to ECAD, minus the writing/reading of a baseline) are expected to happen numerous times in the design process, sending change files and response files to each other until the design in finalised.

You will read down the 'column' and the column of commands used will depend on your starting point – from Pulsonix or from the MCAD system. Both workflows are summarised below:

Pulsonix PCB to MCAD

Write Baseline provides a starting point and is the first file written. This could be considered the master file from which all other changes are based.

Run **Find Changes** to inform you of changes made to the design. At this point you can move on to **Write Changes**, or:

You may decide to make further PCB changes to your PCB design.

Back in the **IDX Collaboration** dialog, you will run **Find Changes** again to read latest changes.

This can be an iterative process.

At some point, you may decide to commit the changes and will **Write Changes** to a changes file.

In the MCAD system, **Accept Changes** and write a response.

Back in PCB - **Read MCAD Response**. At this point a Clearance file is automatically generated. It acts as a 'receipt' that both designs have processed the change.

Note: This is not necessarily present in all implementations of IDX.

If MCAD has made changes you do read them in using Read MCAD Changes, and this will automatically populate the grid with these changes (the Find Changes button will not be used, as this finds the changes between your current PCB design and the current IDX Baseline file).

The MCAD to Pulsonix PCB Process

You have want to start by creating your board outline and other critical items required in your MCAD system. Your board must fit the enclosure size for example. You may also need to create mounting holes to fit and place critical components etc.

You will now write an IDX baseline file that can be ‘pulled’ into your PCB to start the design or add information to an existing design.

In Pulsonix, in the **IDX Collaboration** dialog you will select **Read MCAD Baseline**

Once a change file has been read in, the grid is populated with all the changes present in the file.

You can use the **Show Change** button on a selected item to highlight the item in the design.

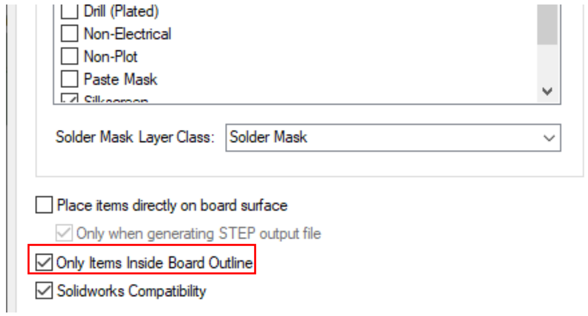
The 'Response' column is now visible in the grid, and allows toggling between accepting and rejecting each change. (Note: You can use the 'Show Change' button, or double click on a grid row to show the selected change in your design).

Once you are happy with the accept/reject state of the changes, use the **Write Response** option to generate an IDX response file for the changes. This should be read in by MCAD, which will accept/reject the changes in their design, so that both designs are synchronised.

3D Viewer Changes

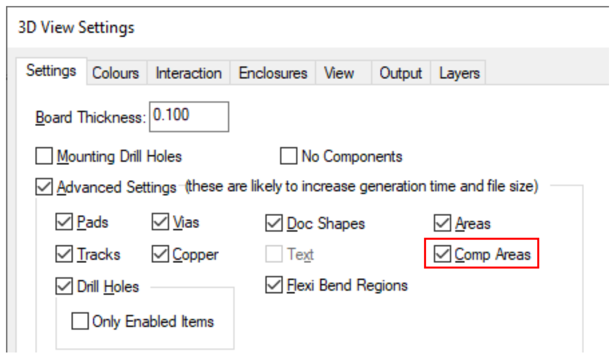
Only Items Inside Board Outline option

Added **Only Items Inside Board Outline** option to **3D Settings, Settings** page. When enabled, any items that are fully outside the extents of the board outline will not be included in the 3D Viewer or output to STEP. Any items inside the board outline, or overlapping the board outline will still be included as normal.



3D Viewer includes Component Areas

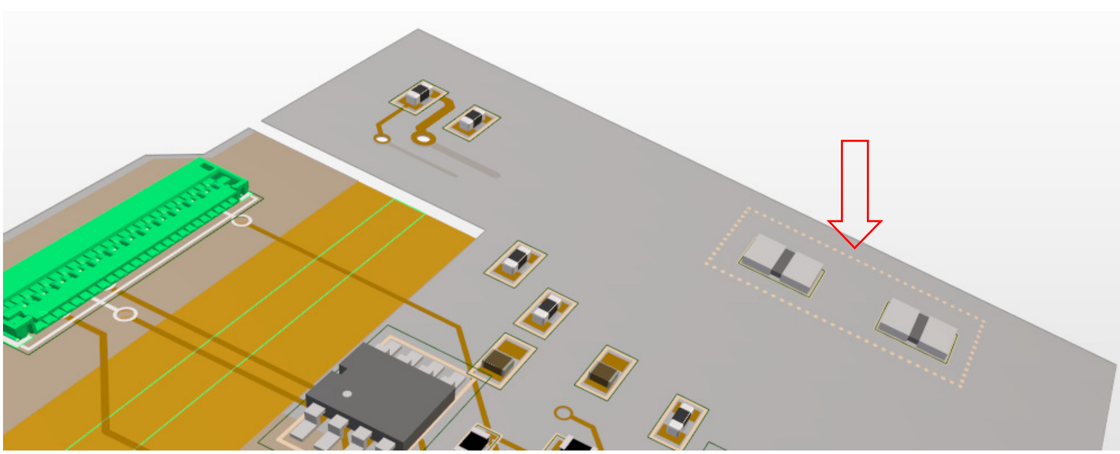
A **Comp Areas** check box has been added to the **3D Settings** dialog on the **Settings** page. This option works similarly to the **Areas** check box, but will cause areas defined in Component footprints to be displayed.



This feature was back-fitted to V12.5.

View of Dotted/Dashed Line Styles

Segments with dotted/dashed line styles are now displayed in the 3D Viewer. Previously, these segments were added as solid lines.

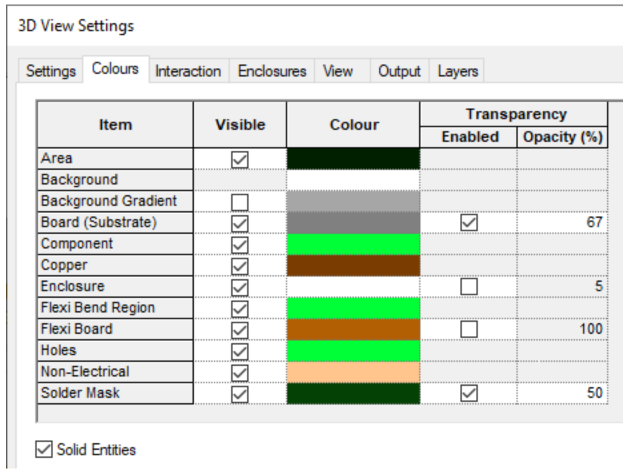


Currently, only Component and Via Keep in and Keep out Areas are shown in the 3D Viewer and only when the Area check box is enabled in the 3D Settings dialog.

Changes to 3D Settings Colours Page

Colours Switches

The **3D Settings Colours** page no longer uses combo boxes with named colours. This has been changed to mirror how colours are displayed in the Colours dialog.



Item	Visible	Colour	Transparency	
			Enabled	Opacity (%)
Area	☑			
Background				
Background Gradient	☐			
Board (Substrate)	☑		☑	67
Component	☑			
Copper	☑			
Enclosure	☑		☐	5
Flexi Bend Region	☑			
Flexi Board	☑		☐	100
Holes	☑			
Non-Electrical	☑			
Solder Mask	☑		☑	50

There is now a grid, populated with the different types of 3D items that can have their colours changed. All items are shown in alphabetical order.

The **Gradients** switch is now incorporated into the grid under **Background Gradient** with the colour setting next to it.

A **Visible** column has been added to the grid this page. Toggling the check box for an item will show/hide all items of that type in the 3D Viewer (e.g. disabling Non Elec visibility will hide all non-electrical items), without the need to regenerate the entire view. This does not affect which items are output when using the Output STEP or Output STL options; it only affects which items are currently shown in the viewer.

In the **Colours** column, you can change the colour of the specific item using colour wells (rather than named colours).

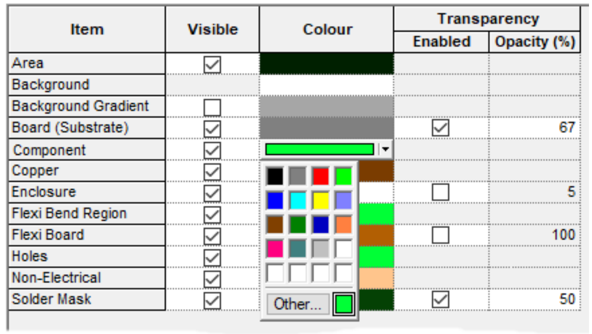
The **Transparency** of certain items can be **enabled**, and will need an **Opacity %** value in the range from 0 to 100% (0 being fully transparent and 100% being opaque).

Outside the grid, the existing check box for toggling if objects are **Solid Entities** or wireframe is still available.

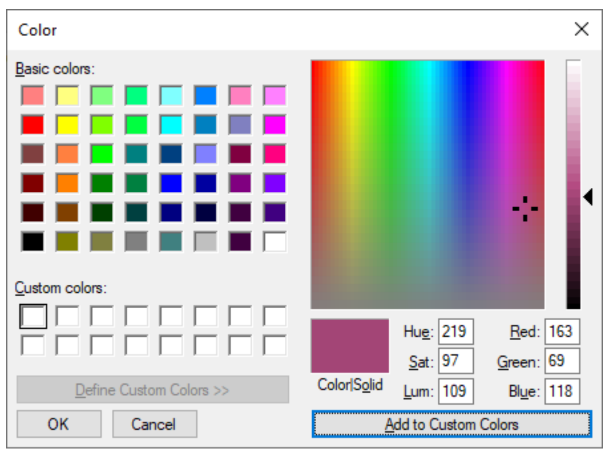
Colour Palette

The colour palette has now been changed to standardise on the windows style palette.

To change colours for any of the categories shown on the dialog, simply click on the drop-down list alongside the category name, and choose the colour you wish to use.

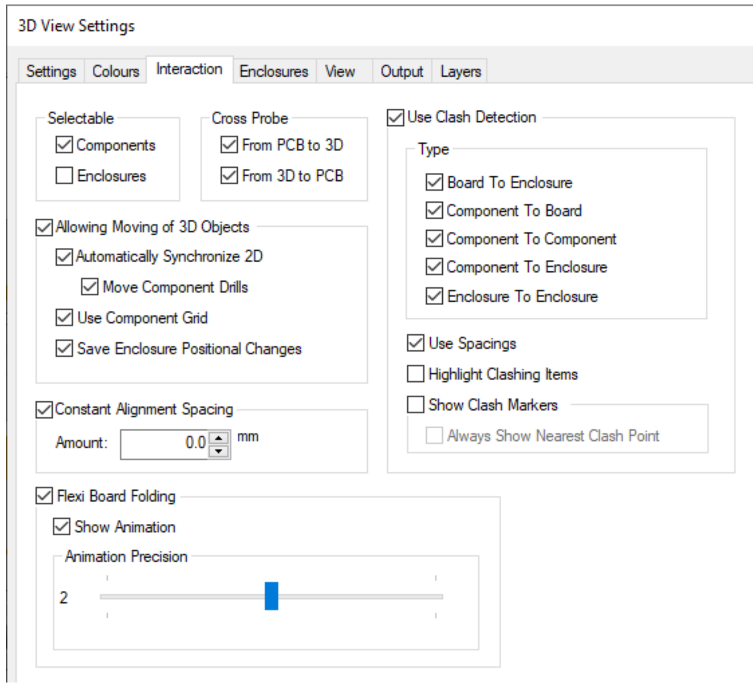


If you want to use a colour that is not shown on the list, choose the 'Other..' entry at the foot of the list, to access the standard Windows colour picker dialog.



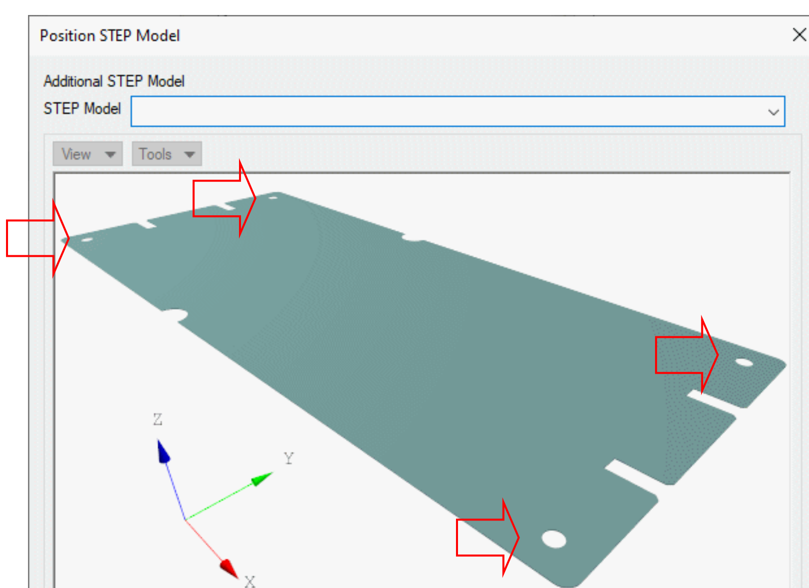
Changes to 3D Settings Interaction Page

The **3D Settings, Interaction** page has been updated and rationalised. The check list boxes used for selectable items and clash detection have been removed and replaced with individual check boxes for each option. The dialog has been reorganised to account for this change.



Enclosure Position STEP Model dialog – Mounting Hole Drills

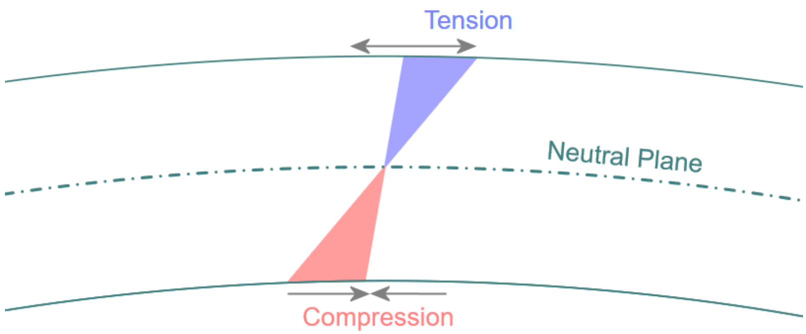
The **Enclosure Position STEP** model dialog, located in the **3D Settings**, **Enclosures** page after clicking the **Add** or **Edit** button, contained a board without any drills. This board will now contain any mounting hole drills that are present in the design. This is useful for the alignment of enclosures.



This feature was back-fitted to V12.5.

Flexi Folding – Neutral Bending Axis

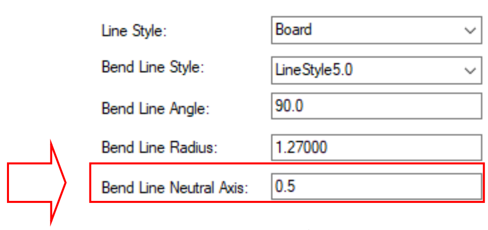
The **Neutral Axis** is a plane within the surface of the shape, such that when the material is bent, the plane does not undergo any tension or compression, and so is the 'true' length of the bent shape. The image below illustrates the stress distribution:



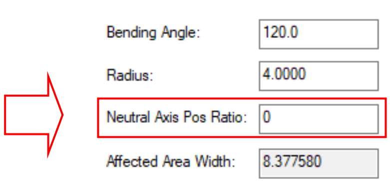
The position ratio is a real number, from 0 to 1, that defines how far through the flexi board the Neutral Axis is (with 0 being the Outer edge of the bend, and 1 being the inner edge). This ratio alters the size of the bend region that is created, for a more accurate calculation of the Affected Area Width (this value can be seen in **Bend Line Properties**). The default value for this ratio is 0.5.

Defining the Neutral Axis Value

This value can set by default using the **Design Settings**, **Defaults**, **Board** page:

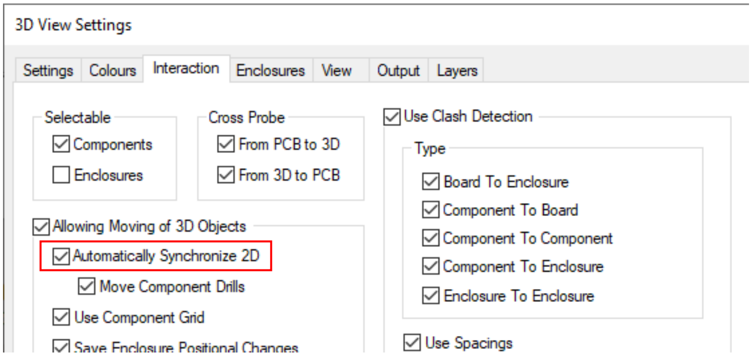


It can also be on each **Bend Line** by using the **Properties** dialog:



Performance Enhancement For Attached Items After Moving A Component

With the **Automatically Synchronise 2D** check box enabled in **3D Settings, Interaction**, if a component is moved in the **3D Viewer**, attached items are updated to their current state in the PCB design view. This removes the need to completely regenerate the 3D viewer if you want attached tracks to be updated.



Performance Update on Move Drill Holes after Moving Component

A new option - **Move Component Drills,** has been added to the **3D Settings, Interaction** page under the **Allowing Moving of 3D Objects** option. This option is only available when **Automatically Synchronise 2D** is enabled. When enabled, if a component is moved in the **3D Viewer**, drill holes are moved from their original position to the current position of the component. This removes the need to completely regenerate the 3D View if you want the drill holes to be updated.

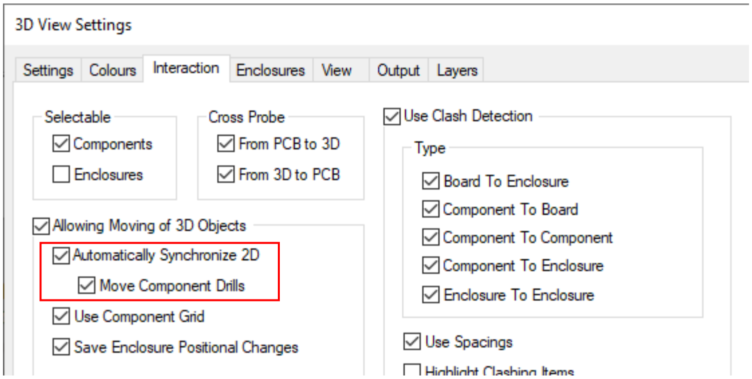


Photo-realistic Display in 3D Viewer

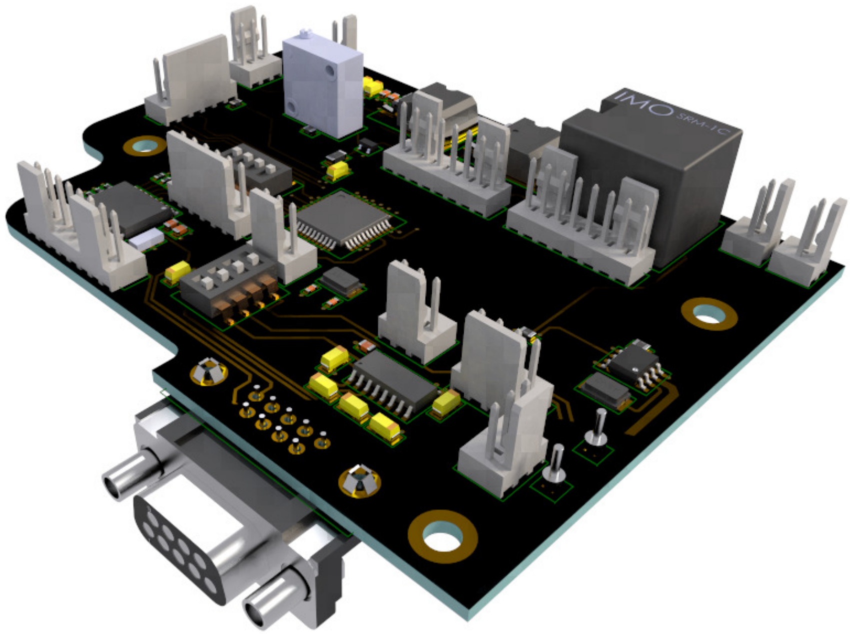
Ray-Tracing and **Path-Tracing** support have been added to the **3D Viewer**. These provide a more photo-realistic display of your design.

Ray-Tracing is a rendering technique that simulates the way light rays propagate in an environment. It involves tracing the path of light rays as they interact with objects in a scene.

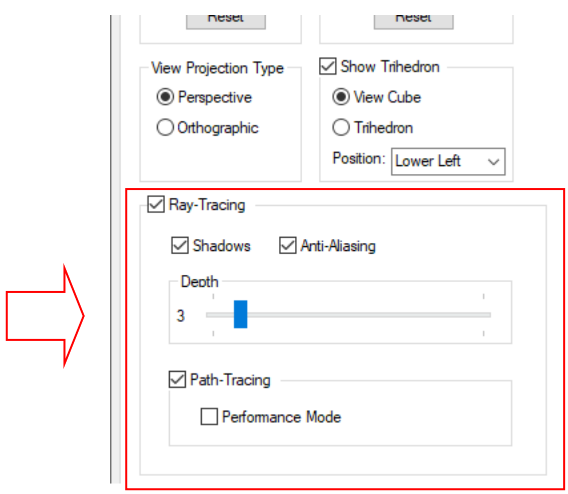
Whereas **Path-Tracing** is a more advanced form of ray tracing that simulates the path of light rays as they bounce around the scene.

While both ray tracing and path tracing aim to simulate the behaviour of light in a virtual environment, path tracing is a more comprehensive and computationally expensive technique that considers a broader range of light interactions, resulting in more realistic images.

Typically, both of these algorithms are more GPU intensive when enabled. If using these modes on large designs, then we recommend a higher performance graphics card is used.



The 3D Settings, View dialog now has additional check boxes for these two modes:



Ray-Tracing

With Ray-Tracing enabled, additional check boxes are available:

Shadows - Enables/Disables shadows rendering

Anti-Aliasing - Enables/Disables adaptive anti-aliasing

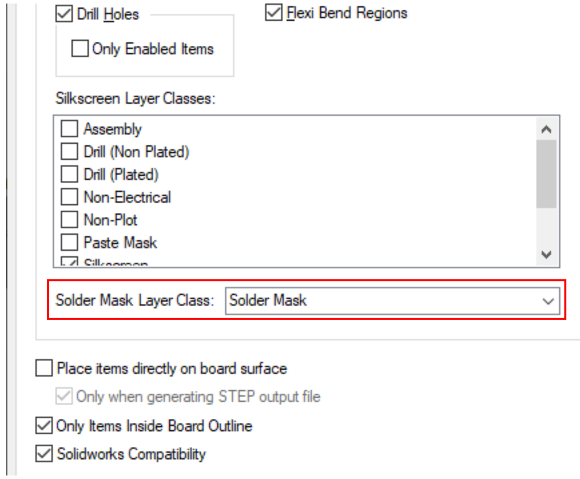
Depth - The number of times a ray can bounce or interact with surfaces in the scene before it is terminated.

Path-Tracing

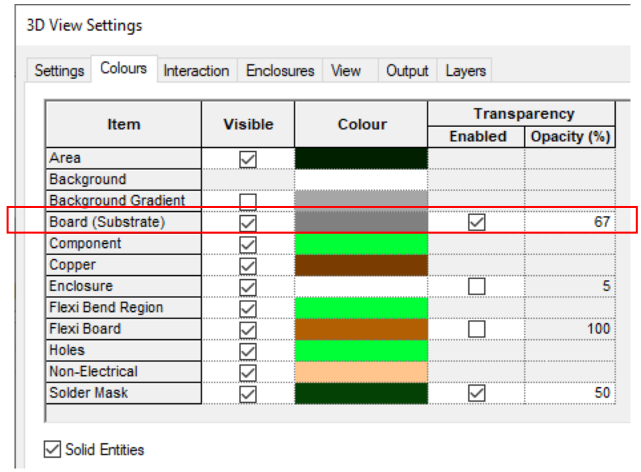
The **Performance Mode** option is a more GPU intensive version of path tracing, to be used if the default version is causing performance issues.

Realistic Solder Mask in 3D Viewer

Within **3D Settings** and **Settings,** there is now a selection for choosing a **Solder Mask**. Solder mask shapes are created for layers of the chosen class, using the solder mask shapes shown in the 2D PCB view as cutouts in the solder mask.



These shapes have their own colour and transparency, defined in **3D Settings, Colour** page.



Single Entity Merge File Size Reduction

File sizes of the **STEP export files** when using the **Single Entity Merge** option (on the **3D Settings Output** page) has been reduced. Reductions of around 50% on average have been experienced.

3Dconnexion SpaceMouse – Background Pulsonix Commands Support

There is now background support for adding Pulsonix commands to the SpaceMouse configuration software so that commands such as **Insert Trac**k or **Properties** for example (but not limited to), can be assigned to a specific button on the SpaceMouse.

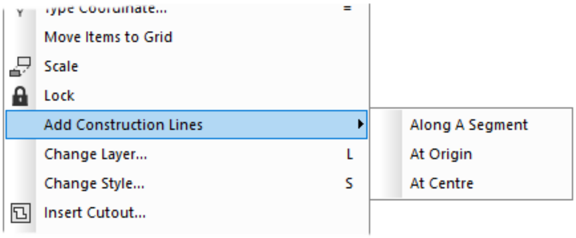
Note, in order for this feature to work, a configuration file must be provided to you by us. The file would contain a list of available Pulsonix commands required. As the commands require a unique ID, you will need to request the commands you want to assign to the keys. Only the command name is required, not the key it will be assigned to. The Pulsonix command list runs into many hundreds, many of which are rarely used. Hence why it is a request-only feature for now.

In order to obtain the commands you require and instructions on how to implement them, you must contact your local support centre and they will assist you.

Construction Line Changes

Add At Centre or Origin

For a selected shape, you can now add **Horizontal and Vertical Construction Lines** at the **Origin** and **Centre** using the two new options from the context menu.



At Origin will pick the origin from where the item was created.

At Centre will use the centre of the item or the centre of the bounding box if the item is an irregular shape.

Select Next when using Along A Segment mode

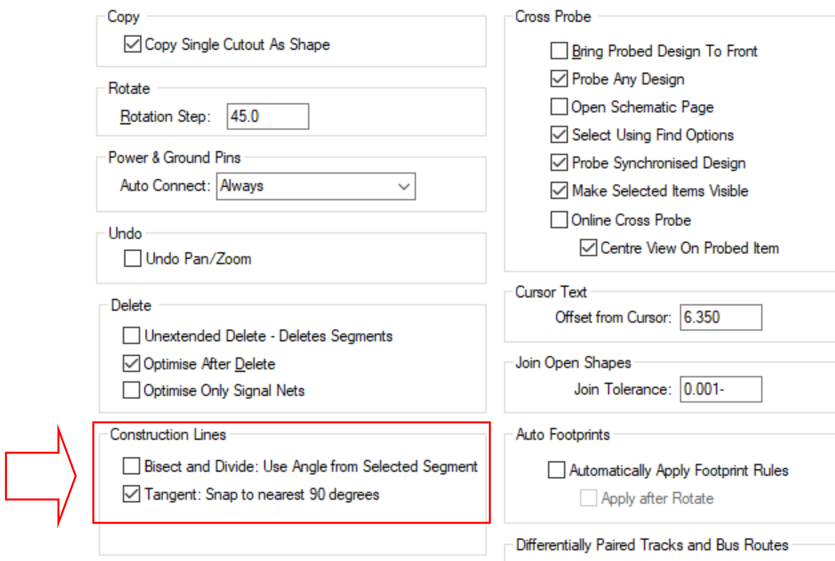
When a Construction Line is placed using **Along A Segment** press **N** (**Select Next**) to jump to another item nearby as with normal select next. Construction line will be moved to be along the next selected segment. This function also works with the **Snap to Edge** option. If a shortcut key is defined, you can also use the Select Previous option

Bisect, divide and Tangent Snapping added to Options Dialog

Two new options have been added to the **Options**, **Interaction** dialog to support construction Line functionality:

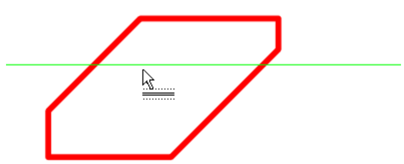
Bisect and Divide: Use Angle from Selected Segment

Tangent: Snap to nearest 90 degrees

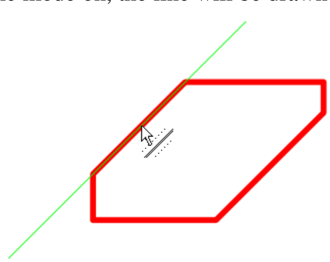


Bisect/ Divide: Use Angle from Selected Segment – Select this option to default to using the angle of the segment if a point on a segment is selected.

With the mode off, the line will be drawn horizontally or vertically:



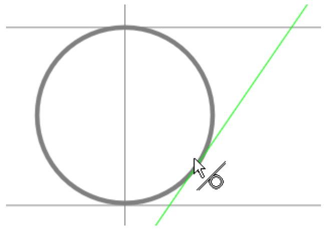
With the mode on, the line will be drawn along the segment selected:



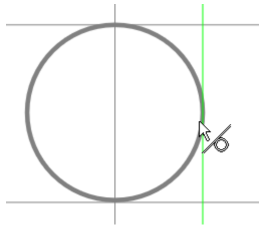
Bisect/ Divide now has line so you can see where the line being placed is before completing the action.

Tangent: Snap to nearest 90 degrees, with this option selected, when using Tangent mode, the Construction Line will be added at 90 degrees rather than wherever the cursor is in free movement mode.

With this mode off, a tangent line will be drawn as normal:

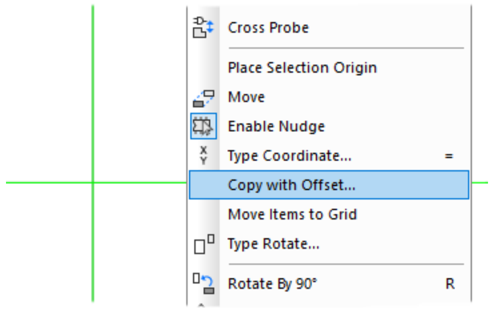


With this mode enabled, a tangent line will be drawn as orthogonal (horizontal/vertical):

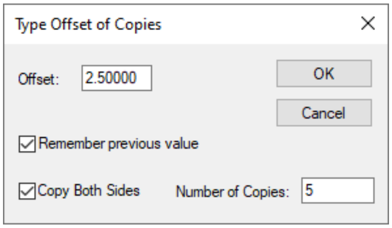


Copy with Offset

When one Construction Line or two perpendicular Construction Lines are selected, the **Copy with Offset** option is available on the context menu.



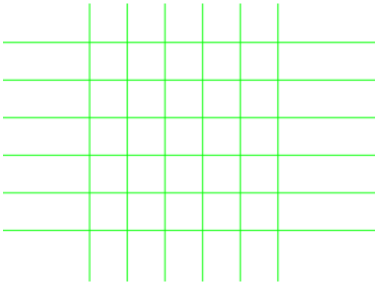
This allows parallel lines to be created at a specified offset from the original selected line.



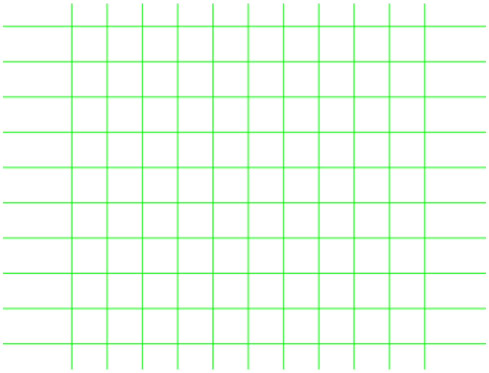
The **Offset** is the distance that the line or lines will be placed relative to original line. The offset copy will be placed right of the lines if a positive number is used and left if a negative number is used.

When used with 2 selected lines, it can form a grid of equal spacing using the lines by selecting the **Number of Copies** to use.

The resultant copy looks like this:

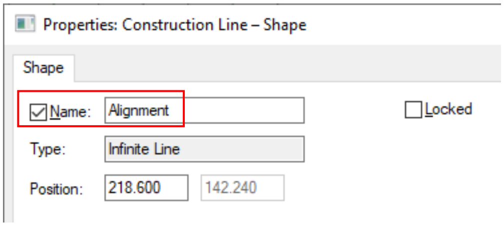


If the **Copy Both Sides** check box is selected, it will copy the **Number of Copies** defined to both sides of the original selected Construction Lines.



Construction Line Names Visible

Names for named Construction Lines are now visible in the design when the **Name** check box is selected in **Properties**.



Named Construction Lines can be found using the **Find Bar**.

Names Move with Construction Lines

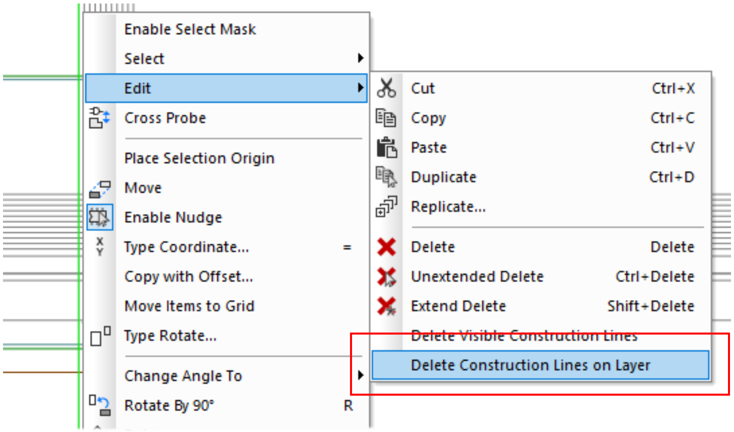
Visible Names for **Named Construction Lines** now move with the Construction Line.

Names are added at the centre of the line, and from then its position is remembered if made invisible and visible again. The centre is chosen as the point when you place the line.

Delete Construction Lines on Layer

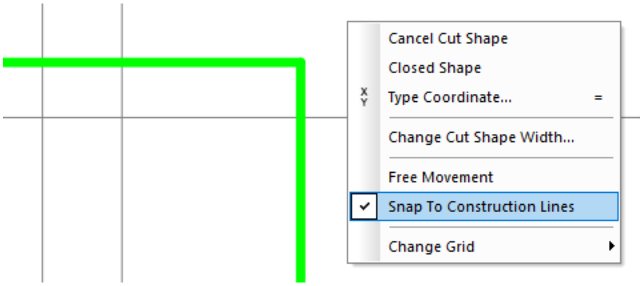
Added option to delete all construction lines that share the same layer as the construction line selected when the context menu is opened.

You now have the option **Delete Construction Lines on Layer** under **Edit** using the context menu with a Line selected in the design. This will delete all Lines that are on the same layer only as the one selected.



Cut Shape – Snap To Construction Lines

When using the **Cut Shape** option, a new context menu option, **Snap To Construction Lines,** is now available. This can be toggled on or off for use. This enables you to snap to Construction Lines instead of using a grid. When using a grid, it will snap to points on the Construction Line that are in line with the grid axes.

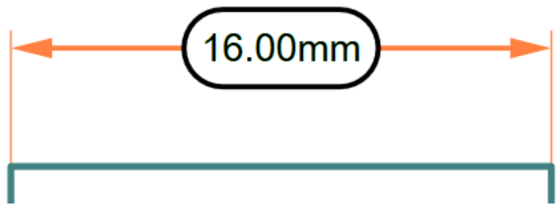


This feature was back-fitted to V12.5.

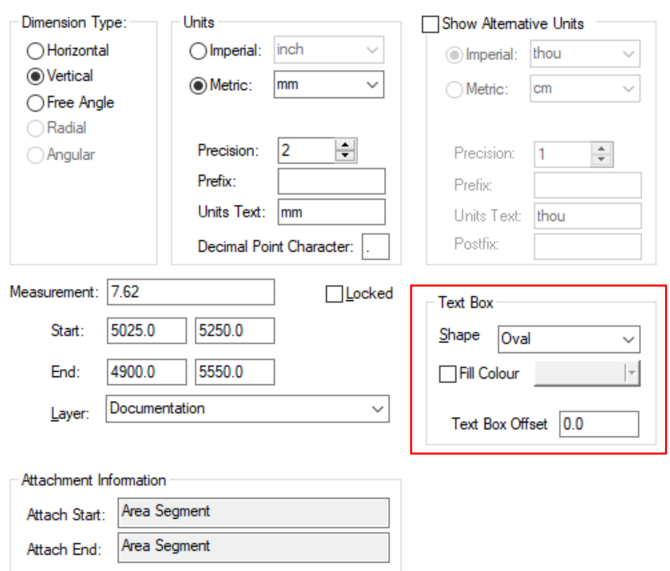
Dimensions

Text Box for Dimensions

Dimension text can now be optionally enclosed with a text box. The text box is dynamic and moves with the dimension as it is placed or resized.

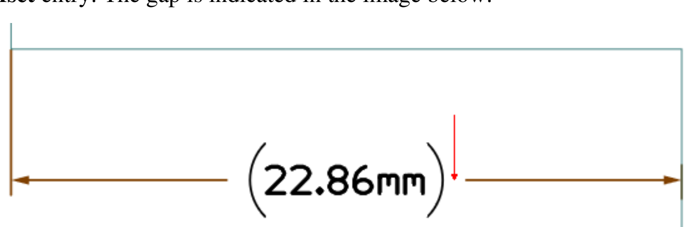


The **Text Box Shape, Fill Colour** and **Text Box Offset** between the text box and lines can be defined in the **Dimension Properties** dialog:

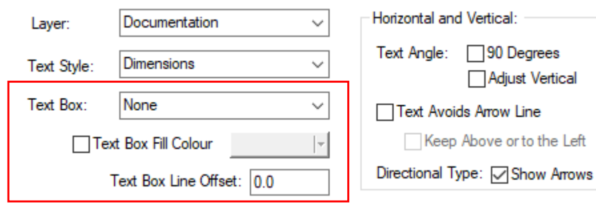


Choose the shape style from the drop down list, **None**, **Rectangle**, **Oval** and **Brackets** ( ).

If you wish to have a gap between the text box and the lines, then specify a value in the **Text Box Line Offset** entry. The gap is indicated in the image below:

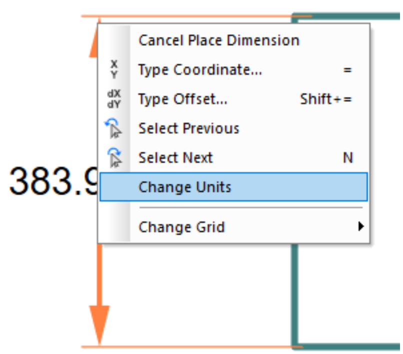


Default values for the **Text Box Shape, Fill Colour** and **Text Box Offset** of the box can also be set using the **Setup** menu, **Design Settings**, **Defaults**, **Dimensions** dialog.



Change Units While Adding/Placing Dimensions

You can now change the **Units** when adding the dimension using the context menu option, **Change Units**. Selecting this option toggles between the units defined in the **Units** dialog.

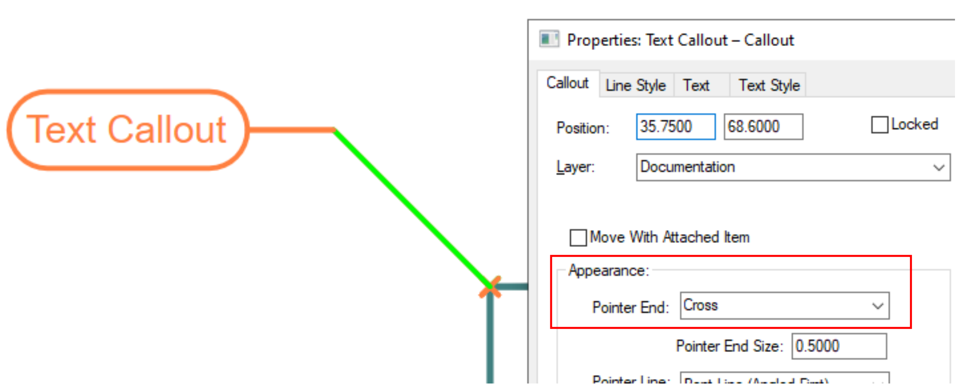


This option is also available when moving the actual dimension text and the dimension line.

Callouts – New Pointer Shapes Cross and Plus

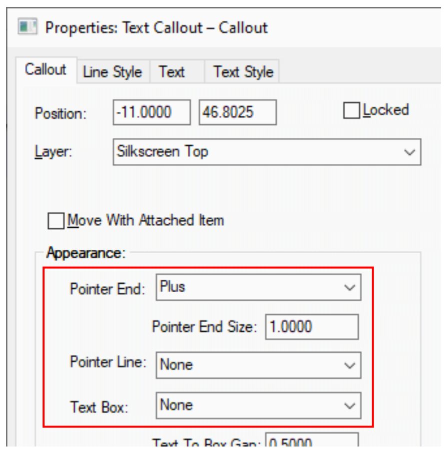
The **Cross** and **Plus** pointer shapes have been added to text callouts for the **Pointer End Appearance**.

For example:

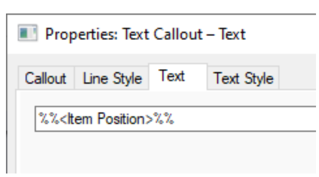


And as a **Plus** shape, illustrated below with an attribute **Item Position** still a callout but no border or line:

Created with a Text Callout using the following Properties:



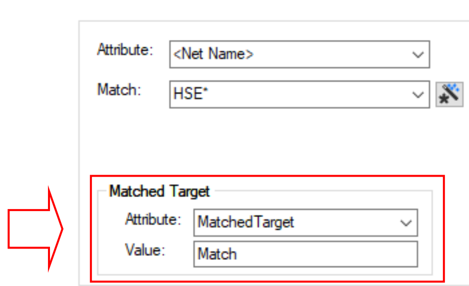
And a Text attribute substitute using <Item Position>:



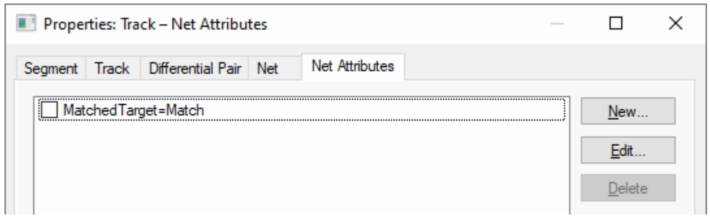
Nominate Net as the Track Length Match Target – High-Speed Option

As part of the **Interactive High-Speed** option, you can now nominate a **Net** to be the **Track Length Match Target** that will be matched in the PCB. You also can still use the existing method of nominating it in the **Rules Spreadsheet**.

The **Matched Target Attribute** and **Value** are defined in the **Technology** under **Track Length Match** rule.

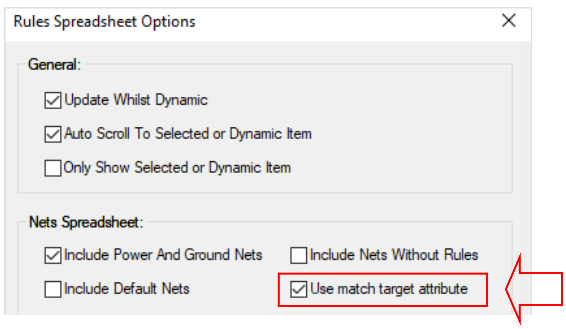


Once assigned, the **Attribute** and **Value** can be attached to a **Net** as a **Net Attribute**. This will then be used in the **Rules Spreadsheet** in the PCB design.

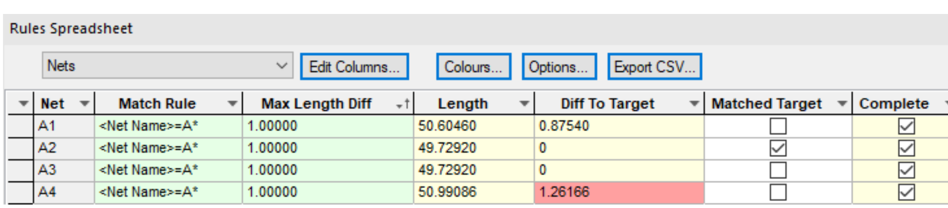


The **Matched Target** attribute can be assigned within a Schematic and passed through to PCB, or assigned directly in the PCB.

In the **Rules Spreadsheet** in PCB, any nets, sub-nets, differential pairs and signal paths that match that rule will be marked as the Target if the option (**Rules Spreadsheet** - **Options**) **Use match target attribute** is selected. When that option is on, you can't modify the **Matched Target** check box, if it is off, your changes (if any) will be saved so switching the option off will restore those changes.



The **Matched Target** entry will show that the attribute has been used and you will not be able to select any other net as the target:



Net	Match Rule	Max Length Diff	Length	Diff To Target	Matched Target	Complete
A1	<Net Name>=A*	1.00000	50.60460	0.87540	☐	☑
A2	<Net Name>=A*	1.00000	49.72920	0	☑	☑
A3	<Net Name>=A*	1.00000	49.72920	0	☐	☑
A4	<Net Name>=A*	1.00000	50.99086	1.26166	☐	☑

Rules Driven Impedance Controlled Routing – High-Speed Option

As part of the **Interactive High-Speed** option, you can now manually route tracks using a track width that is calculated from the **Track Impedance Rule** defined in your **Technology**.

The Process

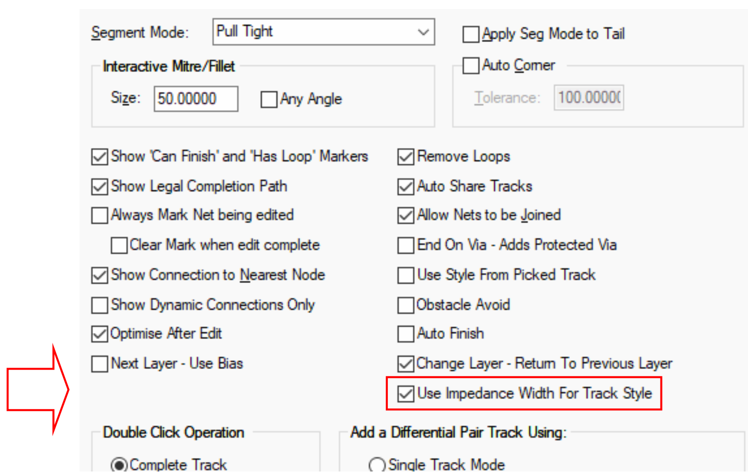
Below is a summary of the process:

- You must have the **High-Speed option** enabled on your license. The HSE feature is a cost option.
- Enable the **Use Impedance Width For Track Style** option in **Options** and **Edit Track** page.

- You must set up a realistic **Layer stack** in the **Layers** dialog along with the actual **Material** thickness to be used.
- Define the **Track Impedance Rules** defined in your **Technology.**
- Start **manual track routing** and the Track Width will be calculated for you.

Enabling Impedance Width For Track Style

To use this feature, you must enable the new option **Use Impedance Width For Track Style** that has been added to the **Options – Edit Track** dialog. Once enabled, other features will then become utilised.



Defining the Layer Thicknesses and Materials

Define a realistic **Layer Stack** in your **Technology** along with accurate **Material thicknesses.** The calculator will not operate correctly if layers are not defined or not positioned in the stack correctly. Each layer must also have a real thickness.

The **Layers** dialog with a real layer stack defined along with Materials and true **thickness**:

	Name	Associated Layer	Class	Side	Bias	Net	Material	Thickness
		Assembly Top	Assembly	Top	None			0.000
Y		Silkscreen Top	Silkscreen	Top	None			0.000
Y	Top		Electrical	Top	X		Copper 1oz (0.035)	0.035
		Solder Mask Top	Solder Mask	Top	None		Mask (0.025)	0.025
		Paste Mask Top	Paste Mask	Top	None			0.000
Y		Pin Names	Non-Electrical	Top	None			0.000
	PrePreg1		Construction		None		Prepreg modified (0.105)	0.105
	GND		Electrical	Inner	Power Plane		Copper 1oz (0.035)	0.035
	Core1		Construction		None		FR4 STD	0.130
Y	Inner 3		Electrical	Inner	None		Copper 1oz (0.035)	0.035
	Core2		Construction		None		FR4 STD	0.130
	Inner 4		Electrical	Inner	None		Copper 1oz (0.035)	0.035
	Core3		Construction		None		FR4 STD	0.130
	VDD		Electrical	Inner	Power Plane		Copper 1oz (0.035)	0.035
	Core4		Construction		None		FR4 STD	0.130
Y	Inner 6		Electrical	Inner	None		Copper 1oz (0.035)	0.035
	Core5		Construction		None		FR4 STD	0.130
	VCC		Electrical	Inner	Power Plane		Copper 1oz (0.035)	0.035
	PrePreg2		Construction		None		Prepreg modified (0.105)	0.105
Y	Bottom		Electrical	Bottom	Y		Copper 1oz (0.035)	0.035
Y		Silkscreen Bottom	Silkscreen	Bottom	None			0.000
		Solder Mask Bottom	Solder Mask	Bottom	None		Mask (0.025)	0.025
		Paste Mask Bottom	Paste Mask	Bottom	None			0.000
		Wires Bottom	Wire Link	Bottom	None			0.000
		Assembly Bottom	Assembly	Bottom	None			0.000
	Construction Lines		Documentation		None			0.000
	Manufacture		Documentation		None			0.000
	Documentation		Documentation		None			0.000

New...
Delete
Delete Unused
Up
Down
Reflect
View Layers...
Only Show Used Entries
Physical Copper Layers: 8
Board Thickness
1.190

The **Materials** dialog defines the **Material Name**, **Thickness** and **Dielectric Constant**:

	Name	Thickness	Electrical Conductivity	Dielectric Constant (ε_r)	Special Use
	Adhesive (0.025)	0.025			
	Copper 1/2oz (0.018)	0.018	595900		
Y	Copper 1oz (0.035)	0.035	595900		
	Coverlay (0.025)	0.025			
	Flexi (0.025)	0.025			
	Flexi (0.05)	0.050			
	FR4 (0.3)	0.300		4.5	
	FR4 (0.4)	0.400		4.5	
	FR4 (1.0)	1.000		4.5	
	FR4 (1.5)	1.500		4.5	
Y	FR4 STD	0.130		4.5	
Y	Mask (0.025)	0.025			
	Plating	0.030+	595900		Hole Plating
	Prepreg (0.05)	0.050		4.5	
Y	Prepreg modified (0.105)	0.105		4.5	
	Thin Plating	0.020+	495900		Micro-Via Hole Plating

Defining the Track Impedance Rules

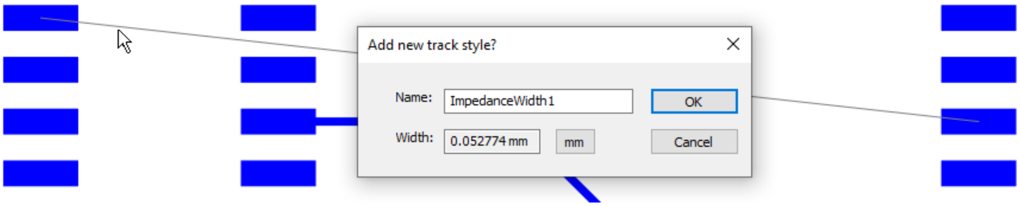
In your **Technology** Define the **Track Impedance Rules** for **Tracks** and **Differential Pairs**.

These rules provide the Impedance required when calculating the Track Width.

Enable	Attribute Name	Match Value	Side	Layer	Area	Single Ended Impedance	Differential Impedance
☑	<Net Name>	CLK				75.000000	<Undefined>
☐	<Differential Pair Name>	DQS*				75.000000	100.000000
☐	<Net Name>	*				75.000000	<Undefined>

Routing using the Track Impedance Rules

With the option **Use Impedance Width For Track Style** selected, when adding a new track into your PCB design, with the other criteria defined, it will it will calculate the **new track width** based on the **Track Impedance Rule** defined in the **Technology**. A new **Track Style** with the new **width** will be offered and the new style added to the **Track Styles** page in your **Technology**.



The default name for the new track style will be **ImpedanceWidthN** (N = 1,2,3…) . Every time a new track style is created, the **Add new track style?** dialog will be displayed enabling you to create the style. This is the default name presented but you can edit it to a name of your own if you wish.

With the **Use Impedance Width For Track Style** option enabled, this overrides any **Net Style** matches defined in your **Technology**.

If a Track Style that matches the **Track thickness** required is already defined in the **Track Styles** dialog in your **Technology**, this will be used.

Once you start routing, if the **Cancel** button is pressed for the calculated, the **Net styles** width defined will be used (if defined, if not, the Default Net Style will be used).

The **mm** button enables you to switch to the alternate units if required. The dialog will automatically present you the new track width in the current design units.

Changing Layers During Impedance Routing

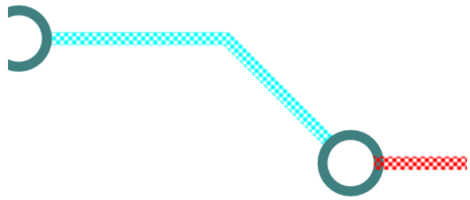
If you change layer when adding a track, a new track width will be calculated based on the **Track Impedance Rules** and **Layer thicknesses** defined for that part of the layer stack.

Option to Draw Tracks Patterned

There is a new option **Patterned Tracks** in the **Options**, **Display** dialog that affects how tracks are displayed on screen. When selected, all tracks will still be drawn in their appropriate colour but instead of being a solid colour, they will use a semi transparent patterned format, similar in appearance to a hatched style used for copper shapes. Other design items underneath the track will be visible through the pattern.

Patterned Stye:

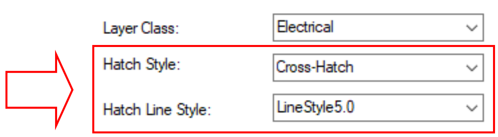
Patterned and Cross-Hatched:



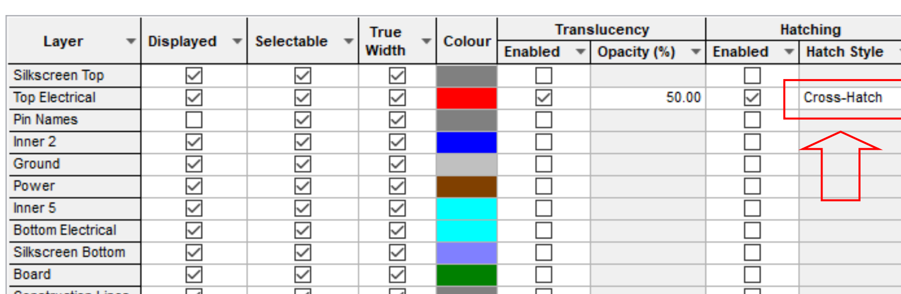
An option has been added to **Options**, **Display**, to enable **Patterned Tracks** and to switch it from a **hatched** pattern to a **Cross** (hatched) **Pattern**.

New Layer items in Layer Defaults

The **Design Settings** option and **Defaults**, **Layer** page has been changed to enable a Hatch Style and Hatch Line Style to be defined.



The **Hatch Style** is the style initially selected when hatching is enabled for a layer. It may be overridden by choosing an alternative hatch style in the **Colours – Layers** dialog.



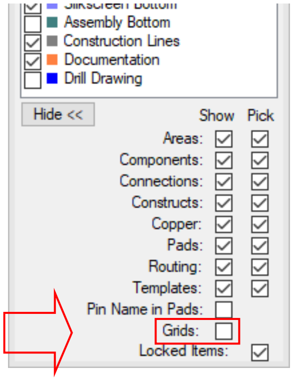
Layer	Displayed	Selectable	True Width	Colour	Translucency		Hatching	
					Enabled	Opacity (%)	Enabled	Hatch Style
Silkscreen Top	☑	☑	☑		☐		☐	
Top Electrical	☑	☑	☑		☑	50.00	☑	Cross-Hatch
Pin Names	☐	☑	☑		☐		☐	
Inner 2	☑	☑	☑		☐		☐	
Ground	☑	☑	☑		☐		☐	
Power	☑	☑	☑		☐		☐	
Inner 5	☑	☑	☑		☐		☐	
Bottom Electrical	☑	☑	☑		☐		☐	
Silkscreen Bottom	☑	☑	☑		☐		☐	
Board	☑	☑	☑		☐		☐	
Construction Lines	☑	☑	☑		☐		☐	

The **Hatch Line Style** is the style used for drawing all layer hatching on screen. For this purpose, it will take preference over the line styles of individual shape items on the layer.

Layers Bar

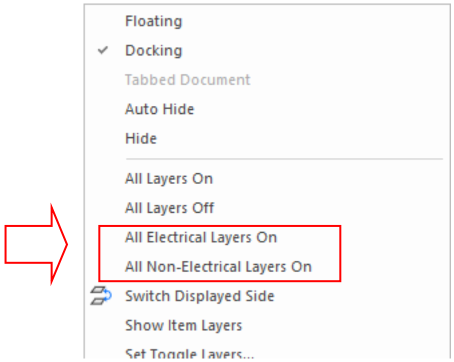
Display Grids button

A check box has been added to the **Layers Bar** allowing easy toggling of the **grids** display.



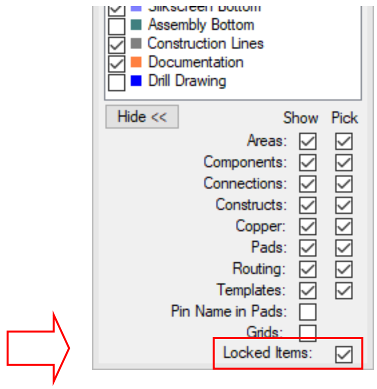
New Layer Selections added

When you right click on the **Layers Bar**, there are two new selections for quickly toggling **Electrical** and **Non-Electrical Layers On/Off**. This is a toggle switch, so once on, the switch enables you to quickly switch them all off.



Toggle Pick Locked Items command added

A new keyboard command - **Toggle Pick Locked Items** has been added. This will toggle the status of the **Locked Items** check box in in the **Layers Bar**. This command can be assigned to a shortcut key if required using the **Customise**, **Keyboard** dialog (from the **Tools** menu) or from the **Edit** menu and **Run Command** option.



<< New command toggles this check box

Layer-based Translucency and Item Hatching

Colours Layers

The **Colours – Layers** dialog now has additional columns for specifying **Translucency** and **Hatching** values for a layer. When specified, all shape-based items such as copper, doc shapes, etc and pads and vias on the layer will appear appropriately translucent and/or hatched on screen.

Layer	Displayed	Selectable	True Width	Colour	Translucency		Hatching	
					Enabled	Opacity (%)	Enabled	Hatch Style
Silkscreen Top	☑	☑	☑		☐		☐	
Top Electrical	☑	☑	☑		☑	50.00	☑	Cross-Hatch
Pin Names	☐	☑	☑		☐		☐	
Inner 2	☑	☑	☑		☐		☐	
Ground	☑	☑	☑		☐		☐	
Power	☑	☑	☑		☐		☐	
Inner 5	☑	☑	☑		☐		☐	
Bottom Electrical	☑	☑	☑		☐		☐	
Silkscreen Bottom	☑	☑	☑		☐		☐	
Board	☑	☑	☑		☐		☐	
Construction Lines	☑	☑	☑		☐		☐	

Translucency

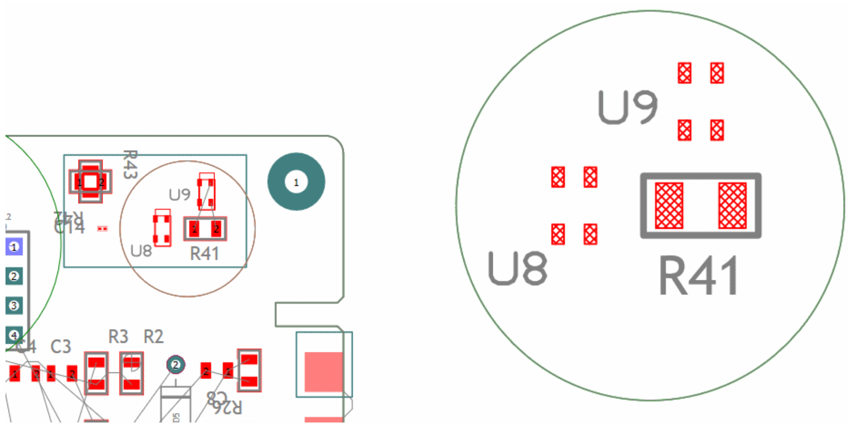
These columns allow a translucency value to be specified that will be applied to design items on the relevant layer. Checking the Enabled cell allows a translucency value to be specified as a percentage in the Opacity (%) cell. By default it is unset, but once set all shape based items such as copper, doc shapes, etc. and pads and vias on the layer will appear appropriately translucent on screen allowing other items underneath to be visible.

Hatching

These columns allows a hatch style to be specified that will be applied to design items on the relevant layer. Checking the Enabled cell allows the Hatch Style cell to be used to choose from those defined in the design's Technology. By default it is unset, but once set all shape based items such as copper, doc shapes, etc. and pads and vias on the layer will be drawn on screen using the chosen hatch style.

Embedded Views - Layer Translucency and Item Hatching

When using **Properties** of an **Embedded View**, the new colours settings for layer translucency and hatching can now be applied to the custom colour settings of the Embedded View. This means you can have different translucency and hatch views of the Embedded View to the main design.



Mark Net – Show Invisible Items

Options dialog

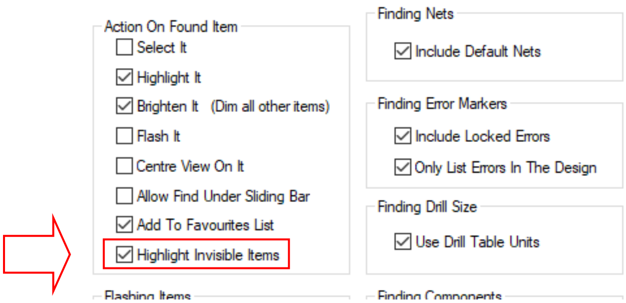
If the **Show Invisible Net Items** option is checked in **Options**, **Display**, then design items on the marked net, that would otherwise not be visible, are displayed highlighted in the Mark Net colour. They will remain visible as long as the net remains marked.

The Latch Mode Mark Net context menu also now has a new **Mark Net Shows Invisible Items** option.

Find - Highlight shows Invisible Items

A new check box in the **Options**, **Find** dialog has been added - **Highlight Invisible Items**

This will enable/disable the **Find** feature, when the **Highlight It** option is enabled, showing found items, that would otherwise not be visible, as Highlighted.



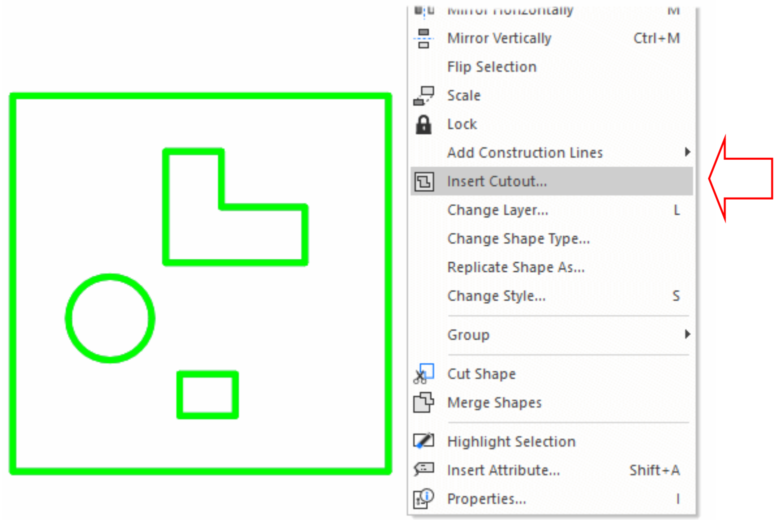
It will also apply to other uses of the **Highlight** functionality.

Invisible Highlighted Tracks Drawn Patterned

Highlighted tracks on non-visible layers are now shown using a patterned style so they are distinguishable from tracks on visible layers.

Add Cutout available with Two or more Shapes Selected

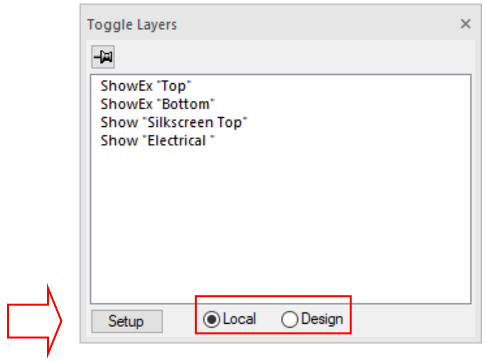
When a shape is selected that contains one or more shapes within it, the **Add Cutout** option now appears on the context menu. The resultant shapes will be added to the main shape as Cutouts.



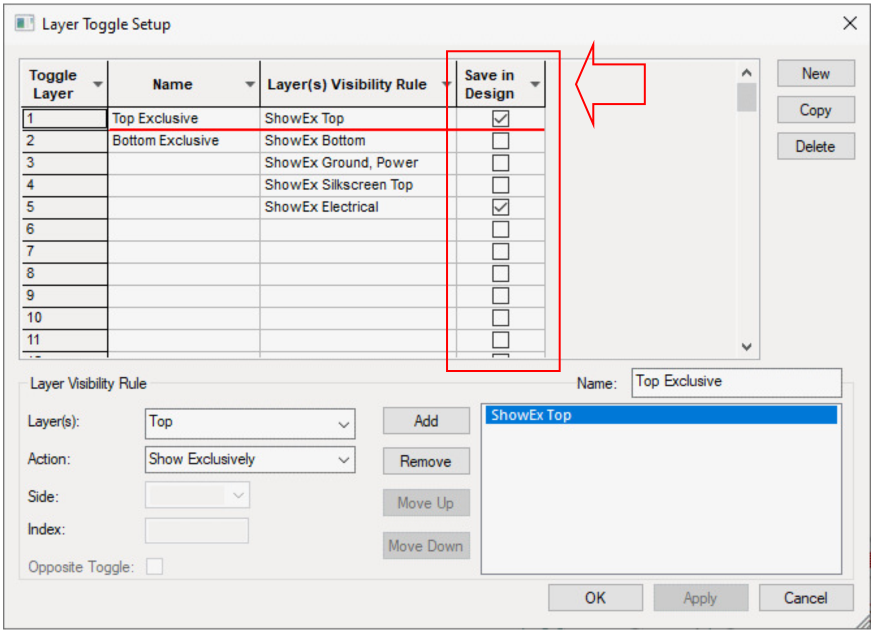
Save Toggle Layer Settings in Design

In the **Toggle Layer bar**, there are now two additional options next to the **Setup** button – **Local** and **Design**. These control how the Toggle Layers settings are used.

This means you can save specific layer combinations where layer names are unique for example, and save them with the design for use by another user. It also enables you to create your own unique local set for your own purposes, and to mix the two sets.



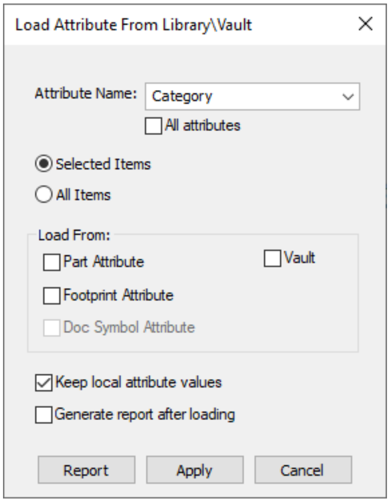
If the **Local** option is selected, then when the **Setup** button is pressed, the dialog will have an extra grid check option **Save in design**. The Toggle Layers that were checked will be visible when the **Design** option is selected.



When the **Design** option is selected, the **Setup** dialog will show the Toggle Layers that were saved in design, the extra grid check boxes (for **Save in design**) are not visible in this instance.

Reload Attributes Only option

A new option - **Load Attributes From Library,** has been added to the context menu when a Component is selected in the design. Within PCB or Schematic designs, use this option to load just **Attributes** only from the Library or Vault into Components in the design. This is an alternative to using the **Reload from Library** option.



You can choose to Reload between a specific Attribute Name that you select from the drop down list, or **All Attributes**. The drop down list is populated from Attribute names of type **Any Item** or **Part** that are found in your **Technology**.

When run, the radio buttons allow you to choose between just **Selected Items** or **All Items** in the design.

When loading the attributes, you can choose which source they will come from using the **Load From:** selections.

Check the **Keep local attribute values** box if you want all attributes currently on your Component to be retained.

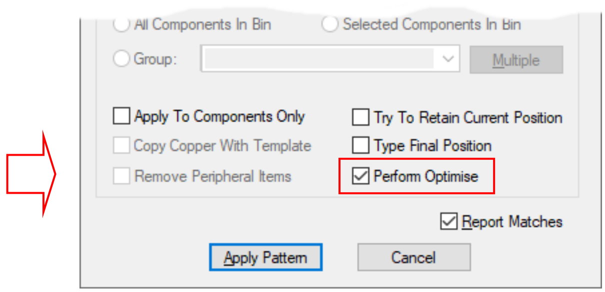
With the **Generate report after loading** button selected, a report of the attribute changes will be displayed once the Apply button has been used.

The **Report** button displays a summary dialog of the items and the attributes that are about to be loaded.

Apply Layout Pattern – Perform Optimisation option

The **Apply Layout Pattern** dialog now has check box for **Perform Optimise**.

This option determines if an optimise nets is performed after the applying the pattern. You can turn this option off on large designs.

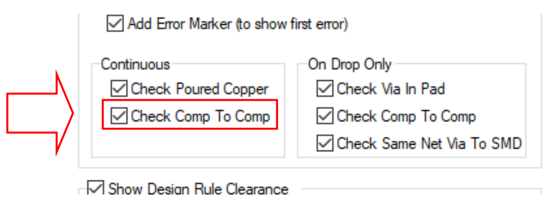


Apply Layout Pattern – Copy Group Patterns from Source File

In the **Apply Layout Pattern** option, it will now attempt to copy existing group patterns from the source to the new pattern.

Continuous Component to Component Checking during Move

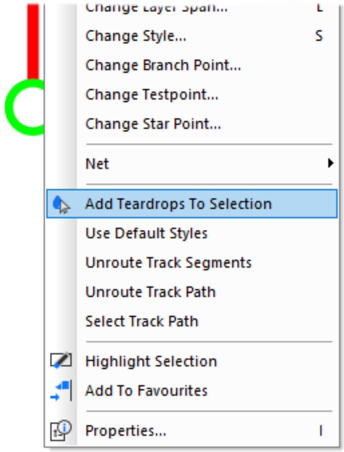
A option in has been introduced to the **Options** dialog under **Online DRC** and the **Continuous** section. Check **Comp To Comp** to stop Components moving too close to other Components during move (if **Continuous** is checked). It draws Component clearances as translucent areas.



This new switch also stops Components from entering Component keep-out areas and draws clearances for these as well.

Add Teardrops to Selection on Context Menu

An option has been added to the context menu to **Add Teardrops To Selection** when appropriate items are selected. The context menu feature behaves in the same manner as selecting it off the **Tools** menu.

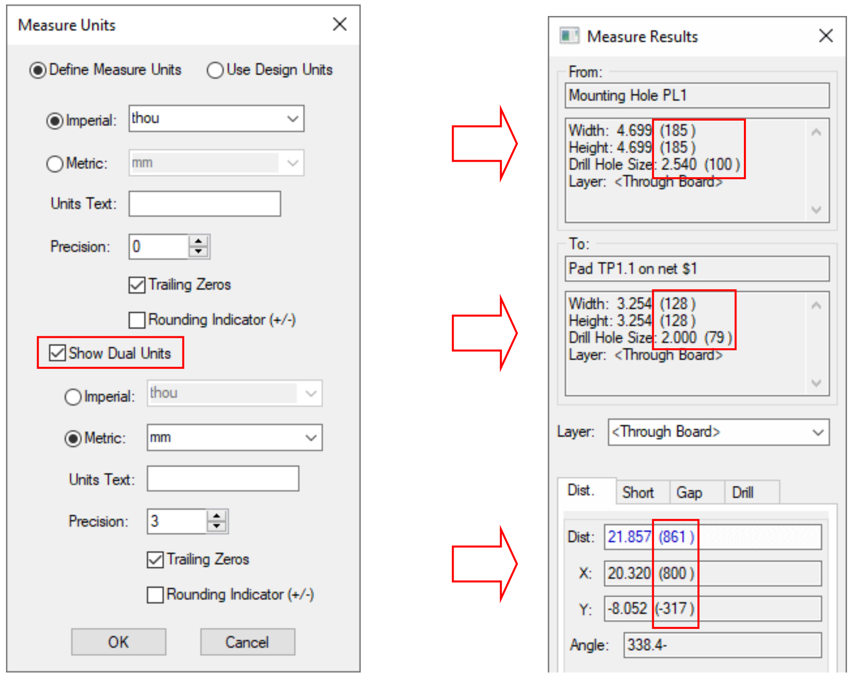


Online DRC for Add Teardrops to Selection

The **Online DRC** options set now only apply to the **Add Teardrops To Selection**, and not any of the 'To Nets' options, which are always checked. This means you can override potential DRC errors that may occur when adding specific teardrops.

Measure Tool - Show Dual Units

A check box has been added to the **Measure Units** dialog to enable you to **Show Dual Units**. When checked, extra controls become available to define the second unit of the measurement, which is in turn, displayed on the main **Measure** dialog.



Changes to Drill Count Reported in Drill Tables and Reports

The drill holes count reported in the Drill Table and any Reports do not count drills in Components in the Component Bin.

(Previously, this reported all drill holes including ones in the Component Bin).

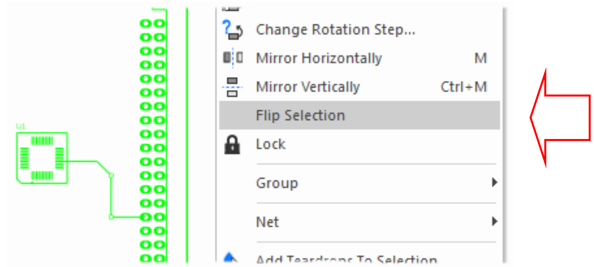
This feature was back-fitted to V12.5.

Drag and Drop for DXF and Gerber Files

You can now drag and drop **DXF** and **Gerber** files from **File Explorer** into Pulsonix. The relevant import dialog will open and the import be actioned onto the currently open design.

Flip Selection Command

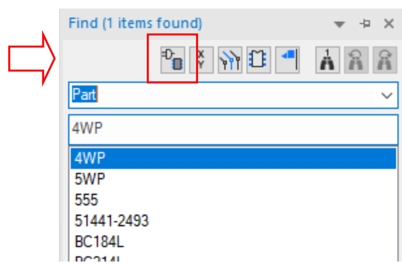
When you have items selected that exist on a layer that has a swap/matching layer defined in your Technology, you can now select the **Flip Selection** command from the context menu. This mirrors and changes layer of all the items selected.



Find

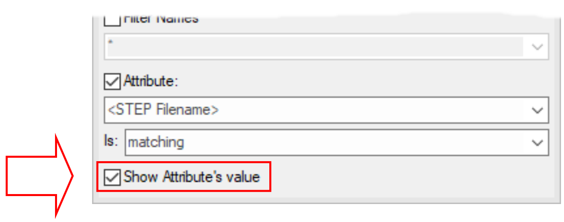
New ‘Quick Access’ Toolbar button for Parts in Find Bar

There is a new shortcut button for accessing **Parts** in the **Find** bar.



Show Attribute Value for Parts

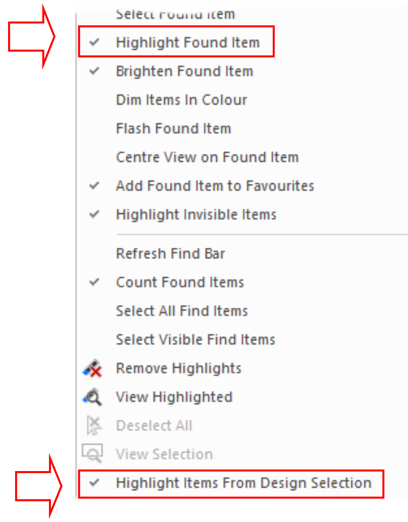
A new option to **Show Attribute Value** when finding **Parts** has been added. When applied, the attribute value is displayed instead of the Part name.



Components in Marked Nets Highlighted in Find Bar

When the **Find** bar is open and the selection is on **Components**, when a **Net** in the design is marked (using **Mark Net**), the Components connected to the net will be highlighted in the Find bar. Once the net has been marked, click in free space to signal the Find bar to update the list.

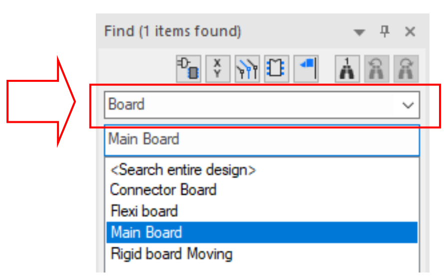
In order for this feature to operate successfully, you must also have the following options set using the context menu in the Find bar – **Highlight Found Item** and **Highlight Items from Design Selection**.



Find Boards

With a **PCB design** open, in the **Find Bar**, you can now select **Boards** from the selection list box to search the design for boards.

Like other named items in Find, boards with names will have their own entry in the list whilst boards without names will share the <No Name> entry. You can cycle between no name boards using the Next button.

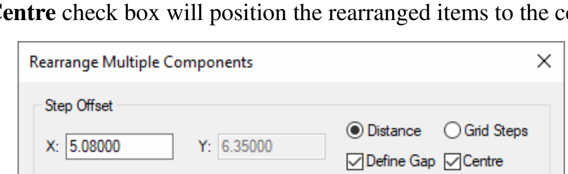


Rearrange Multiple Items - Gap and Centre Options

The **Rearrange Multiple Items** dialog has been enhanced with two additional options – **Define Gap** and **Centre**.

Use the **Define Gap** check box to specify the X Y values a gap between the items.

The **Centre** check box will position the rearranged items to the centre of your design or panel.

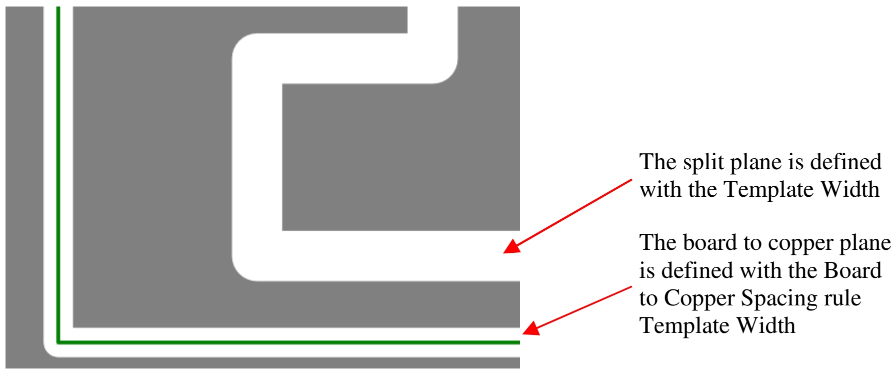


These options are also available in the **Panel Editor** when using **Insert PCB** and choosing **Multiple Items** and when using the **Rearrange Multiple Items** option on selected boards in the Panel.

Change to Split Power Plane Generation

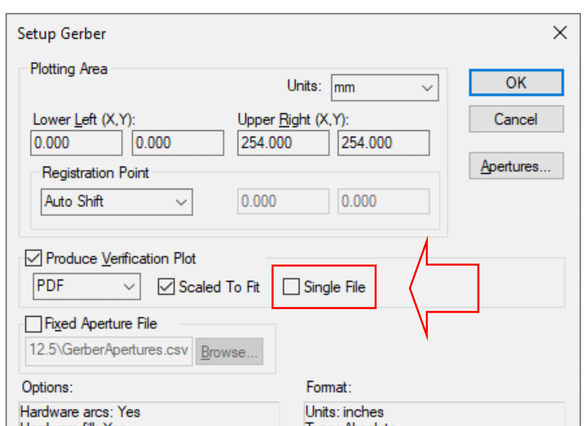
A split power plane now uses the width of the Template to define the gap around the split plane.

Previously, the whole board outline was enlarged if the **Template width exceeded** the **Board to Copper Spacing**. Now the gap for the Board to Copper outside a Template is not dependent on the width of a Template.



CAM Plots - Output Gerber Verification Plots to a Single File

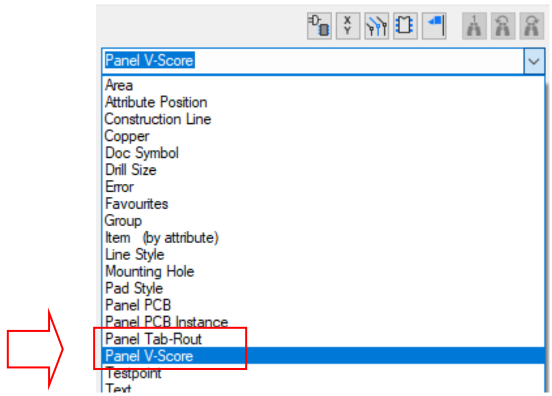
Within the **CAM Plots** option, in the **Gerber Setup** dialog, within the **Verification** section, there is a new option - **Single File**. If enabled, all Gerber verification plots produced will be combined into a single file. This only applies to the Windows PDF output or PDF output.



Panel Editor

Find - Panel V-Scores and Tab Routs

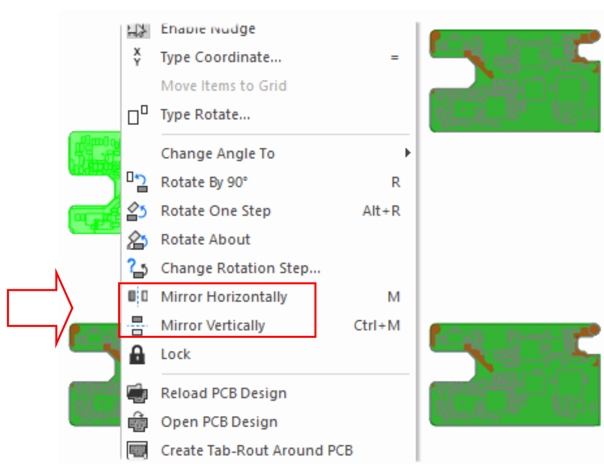
When using the **Panel Editor**, the **Find** function now has options to find **Panel V-Score** and **Panel Tab Routs** items when used.



Mirrored PCB Instances in Panel

When in the **Panel Editor**, there are now two context menu options available to **Mirror Horizontally** and **Mirror Vertically** an instance of the PCB.

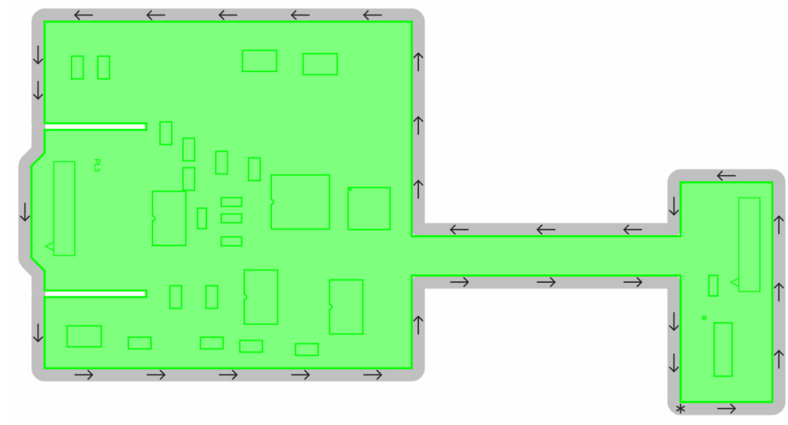
In order to mirror the instance, there must be matching 'opposite' sets of layers in order for the mirror to be performed.



The mirrored instances are represented within the different output formats for panels, such as ODB++ etc. All other Panel features such as inserted tab routs around board instances. work with these mirrored instances also.

Tab Routing for Flexi Board Outlines in Panels

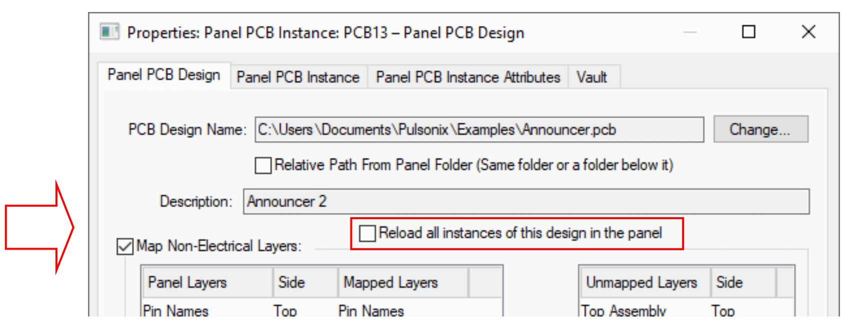
When a design contains multiple board outlines, including spanned layers used for flexi-boards, when the PCB design is included in a Panel, the outlines are now combined to provide one overall shape that can then be used to create a tab routing shape around. When adding a PCB instance to a Panel it converts all overlapping boards to one merged board in the instance.



Update All PCB Instances in a Panel

A new check box has been added in the **Panel editor** when using the **PCB Instance Properties** of a selected design.

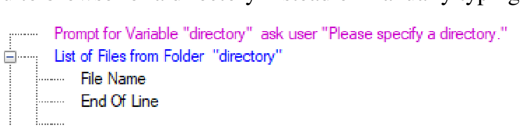
When the **Reload all instances of this design in the panel** box is selected, it will reload all of the PCBs in the panel that share the same PCB as the one which you are on the properties page for.



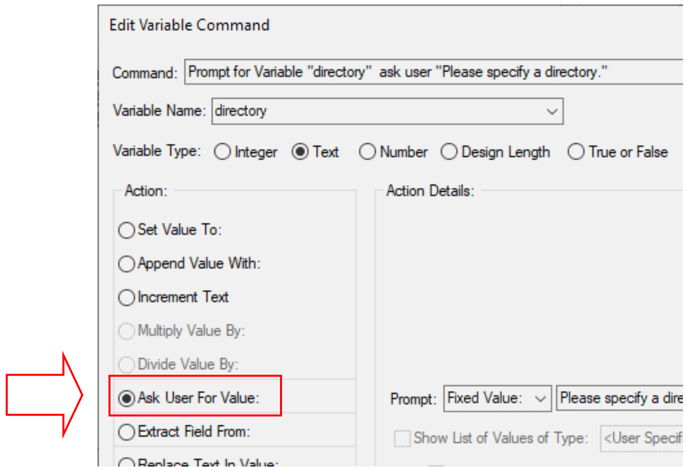
Report Maker Changes

Prompt To Browse For Directory Command

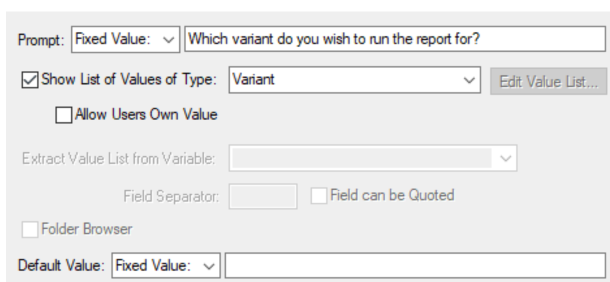
In Report Maker, you can now specify in the **Variable** command, **Ask User For Value**. This will allow you to browse for a directory instead of manually typing in a value when the report is run.



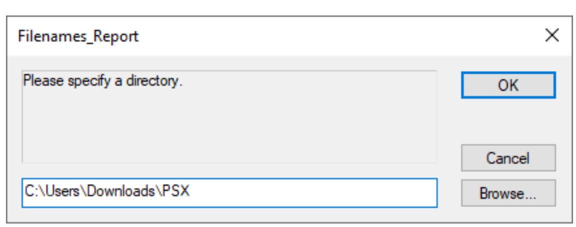
The Variable dialog shows **Ask User For Value**:



To enable this, check the **Folder Browser** box on the **Ask User For Value** section of the **Edit Variable** Command dialog.

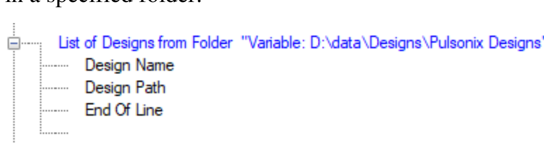


When run, it now prompts you for a folder from which to collect the files:



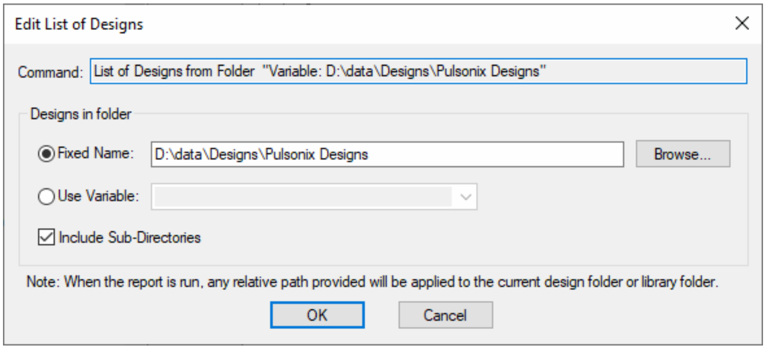
List of Designs Command

A new command, **List of Designs** has been added. This command will allow you to report on all designs in a specified folder.



You can filter which design types you want by enabling and disabling the different **Available For** (PCB or Schematics) types in the Report Maker dialog.

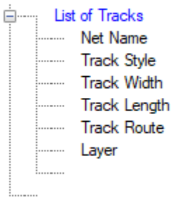
The specified folder can be defined using static text or a variable which can be set by double clicking the **List of Designs** command which will open up the **Edit List of Designs** dialog. If the Vault context is enabled and the report is run from the Vault, the List of Designs command will browse the entire Vault for designs of the specified type.



Once a design is opened, then additional Report Maker commands can be performed on that design, such as writing a list of Parts etc.

List of Tracks Command

A new command, **List of Tracks** has been added. It provides details of the tracks present in the design. It is also accessible under **List of Nets**.



You can use this to obtain a list of segments to get track width along track. Details such as style and end node names also available. Note that track segment end nodes are not available in this interface.

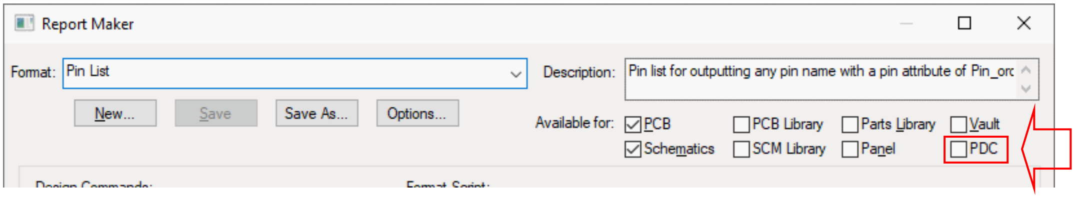
Additional Database Connection Commands (PDC)

Three new commands have been added for use with the Database Connection option (PDC).

In order to run these Report Maker commands, you must have the Database Connection license feature on your license (the PDC is a cost option).

Secondly, the Database must be connected through the Database Connection option on the **Setup** menu. Just having a license and not being connected is not enough to run these commands.

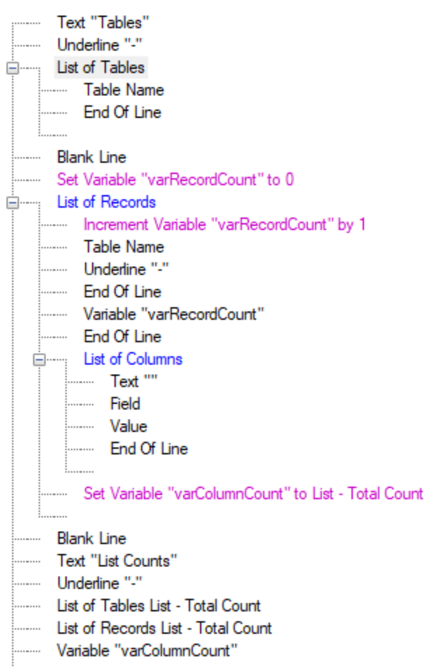
The following are enabled through the **PDC** check box on the Report Maker dialog:



- **List of Tables** - Allows you to report all the tables in the current PDC connection

- **List of Records** - When edited, opens the Edit PDC Command dialog, which allows you create an SQL query. This command with then report all of the records returned from executing that query.
- **List of Columns** - Reports the columns in a List of Records.
  - **Field** - Will report the name of the current field (column)
    Available in **List of Columns**
  - **Table Name** – Will report the current table name
    Available in **List of Columns**, **List of Records** and **List of Tables**
  - **Value** – Will report the value of the current field or the values of all fields if added to **List of Records**
    Available in **List of Columns** and **List of Records**

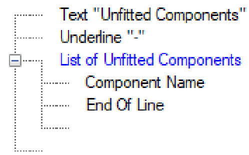
These commands are only available when the new PDC context is enabled in the Report Maker dialog.



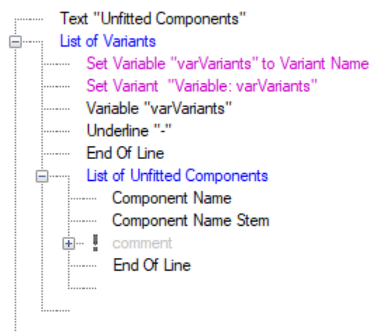
List of Unfitted Components Command

There is a new command in **Report Maker** - **List of Unfitted Components**. This list will provide you the same functionality (and available sub-commands) as the **List of Components** command, but instead of reporting all the components in a design, it reports all the **unfitted components** in the **current variant** in a design.

The basic command would look like this:



The current variant can be changed in the **Setup** dialog under **Variants** and a report run, or by using the **Report Maker** command **Set** Variant then reporting the **List of Unfitted Components** as show in the example below:



List of Areas Command

There is a new command in **Report Maker** - **List of Areas**. This is available at the top level and within **List of Components**.

When in top level, it can be run on any PCB or Schematic design and will list all of the areas and their **Properties**. When used under the List of Components command (**PCB context only**), it will list the areas, and their Properties, in the current Component.

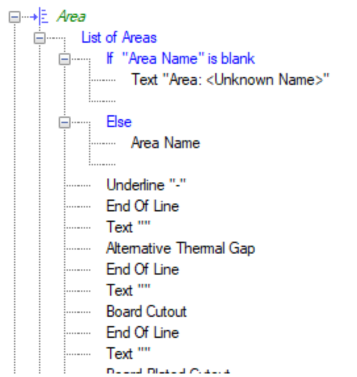
The following commands are available within the **List of Areas** command:

> Alternative Thermal Gap, Board Cutout, Board Plated Cutout, Component Keep, Component Keep Height, Component Pad Keep, Copper Keep Out, Copper Pour Avoid, Drill Keep Out, Micro Via Keep, Mount Hole Keep, Area Name, Power Plane Avoid, Testpoint Keep, Track Keep, Use in Area DRC, Use in Footprint Rules and Via Keep.

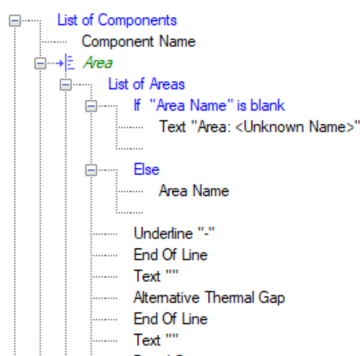
If the **Area Property** is a check box (for example, Board Cutout), the command will report true or false.

If the **Area Property** is a keep (for example, Component Keep) (the combo box containing all the ‘Keeps’), then the selected keep will be reported (Keep In, Keep Out, etc).

The basic command would look like this:



When used within a List of Components:



Track Length Match Target Command

Report Maker can now report the newly added **Nominate a net as the target** features in the **List of Track Length Match Rules**. Within the context of this list command, two new commands are available:

Match Target Attribute – Reports the attribute name that is selected for the rule.

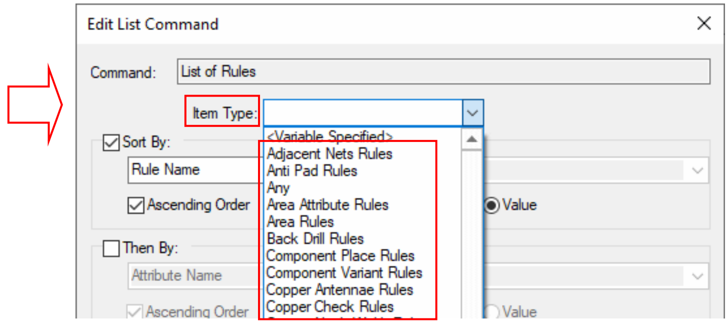
Match Target Value – Reports the attribute value that is specified for the rule.

Component Variants, Areas and Suppress Lands Rules added to List of Rules

You can now report the details of the new **Component Variants Rule** using the existing, **List of Rules** command. This will only be available in a Schematic Design.

Also added are reports for the rule details of **Areas** and **Suppress Lands** using the existing, **List of Rules** command. These will only be available in a PCB design.

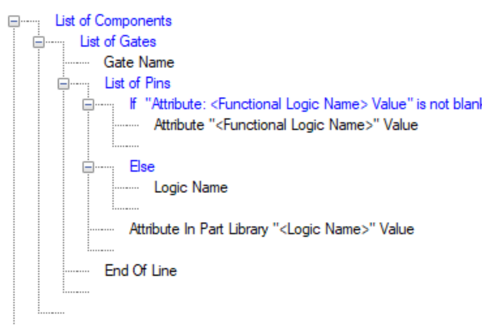
Optionally, you can choose to report one of these rules specifically by selecting it in the **Item Type** box in the **Edit List Command** dialog (by double-clicking the **List of Rules** command).



Pin Attributes in Parts Library Command

You can now use the existing **Attribute In Part Library** command on Component Pins in a PCB or a Schematic design under **List of Pins** for example. It will find the pin in the Part in the library and return the pin attribute.

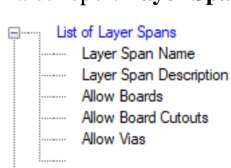
You can use the <Logic Name> attribute to report the logic name from the pin in the Part. This can be used to test against a Functional Logic Name in the design (if one has been used).



Layer Span Usage Commands

Within the **List of Layer Spans** command, new commands are available for **Allow Boards**, **Allow Board Cutouts** and **Allow Vias.** When run, the command will report true or false for the status of the check box for the layer span.

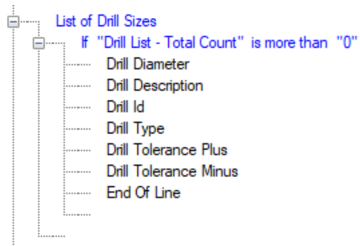
You can also report **Layer Span Description** for layer spans.



Drill Sizes Command Updated

Within the command **List of Drill Sizes**, new commands are available for **Drill Type**, **Drill Tolerance Plus** and **Drill Tolerance Minus** are also available on the **List of Drill Holes**

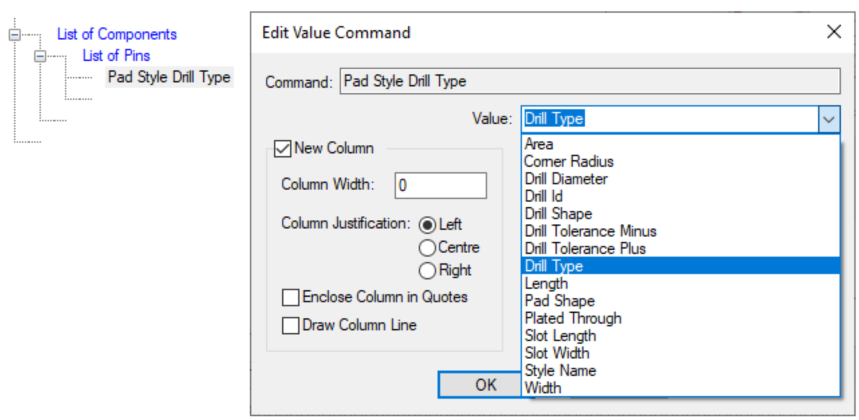
The example below shows the command with a test if any of the drill sizes are used in the design:



The new commands **Drill Type**, **Drill Tolerance Plus** and **Drill Tolerance Minus** are also available on the **List of Drill Holes** command.

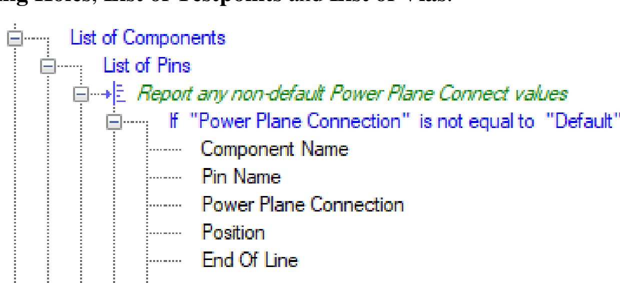
Pad Style Drill Type Command Updated

Within the **Pad Style** command (under **List of Pins**), you can now report the **Drill Type**, **Drill Tolerance Plus** and **Drill Tolerance Minus.**



Power Plane Connection Command

A new **Report Maker** command, **Power Plane Connections**, has been added. With this command, you can report the Power Plane Connection Property that can be assigned to Pads, Vias, etc. This command is available in the PCB context under the **List of Pins**, **List of Free Pads**, **List of Mounting Holes**, **List of Testpoints** and **List of Vias**.



Running this command produces a report like this:

```
Component Pins
--------------
U3 Pin: 1 Not Isolated (1400.00,1650.00)
```

Access attributes in Symbol Library for Footprint in Part Command

The **Attribute in Symbol Library** command is now available from within **List of Footprint Names** within a **List of Parts**.

For example, use this for a Parts library report to list the STEP filename used by the Footprints in each Part.

Vault Update

Using the Vault in V13.0

In order to access the new Vault feature below in Version 13.0, the following changes are required:

Vault Database version

For Pulsonix version 13.0, you should **update** your Vault to version **1007** using the **Vault Setup** dialog. This dialog will also confirm which version you are currently running.

ODBC Driver version

The **V16.0 ODBC driver** is the latest version and will be automatically installed during the main Pulsonix product installation. This version of the driver is required on the Pulsonix client side to support the Postgres server V14.5.

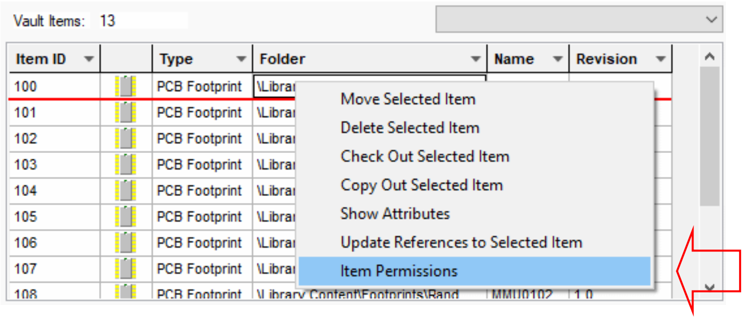
Postgres version

For Pulsonix version 13.0, the PostgresSQL version has not changed and is still **14.5**.

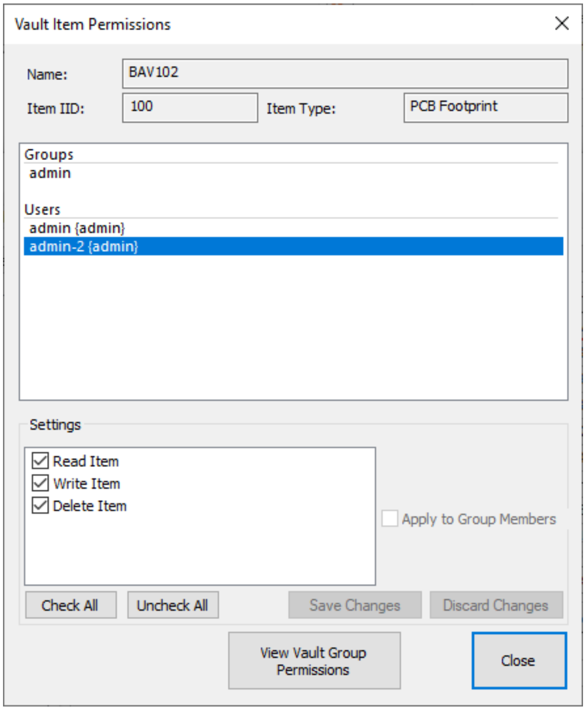
Vault Item Permissions

Permissions can now be set on **Vault Items** and **Vault Folders**.

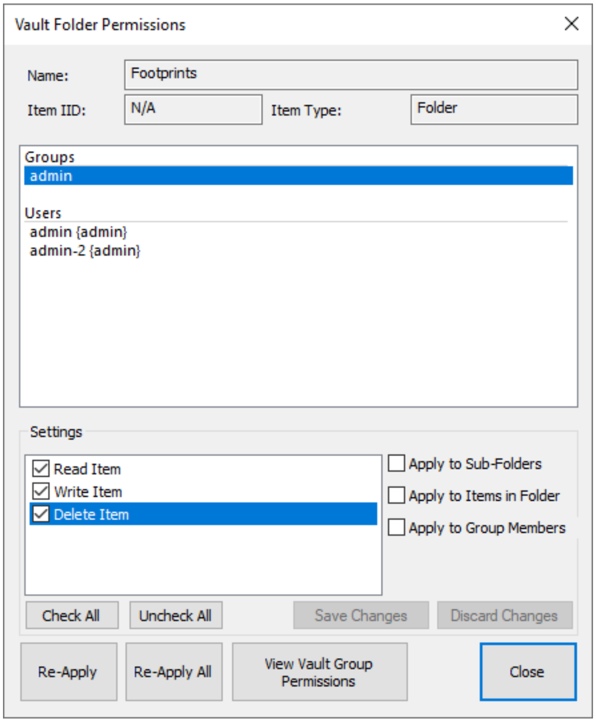
In the **Vault** browser dialog, a new context menu option is available (**Item Permissions** or **Folder Permissions**).



This option is available when right clicking on an item in the Items Grid or a folder in the directory tree. This will open the **Item/Folder Permissions** dialog which allows you to specify Read/Write/Delete permissions for users and groups on that item, folder, items in folder, folders under folder or all of the above.

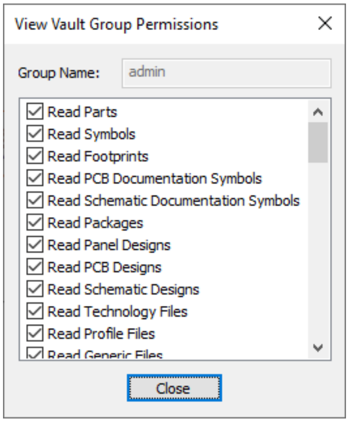


And folder Permissions:



View Vault Group Permissions

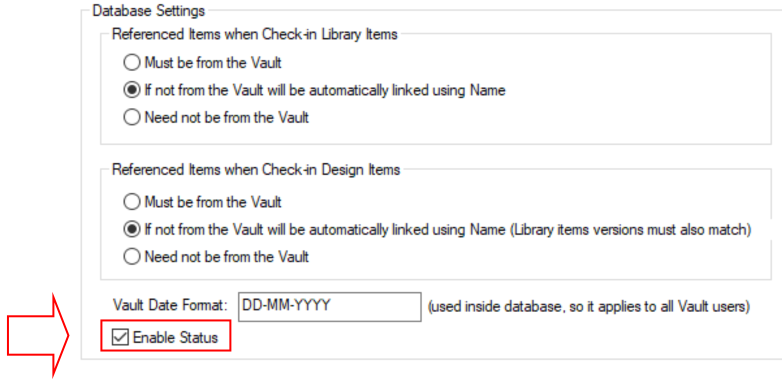
When the **View Vault Group Permissions** button is pressed, you can choose permissions for each **Group Name** and **item** within it.



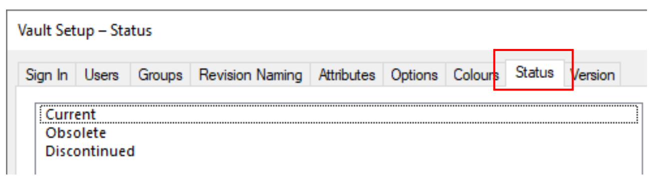
Vault Item Status

You can now create a status that is to be applied to items in the **Vault Browser**.

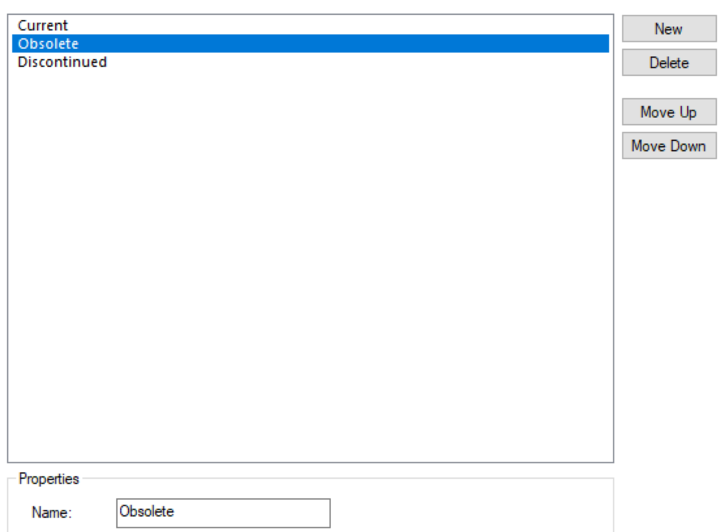
Firstly, in the **Vault Setup** dialog, select the **Enable Status** option on the **Options** page.



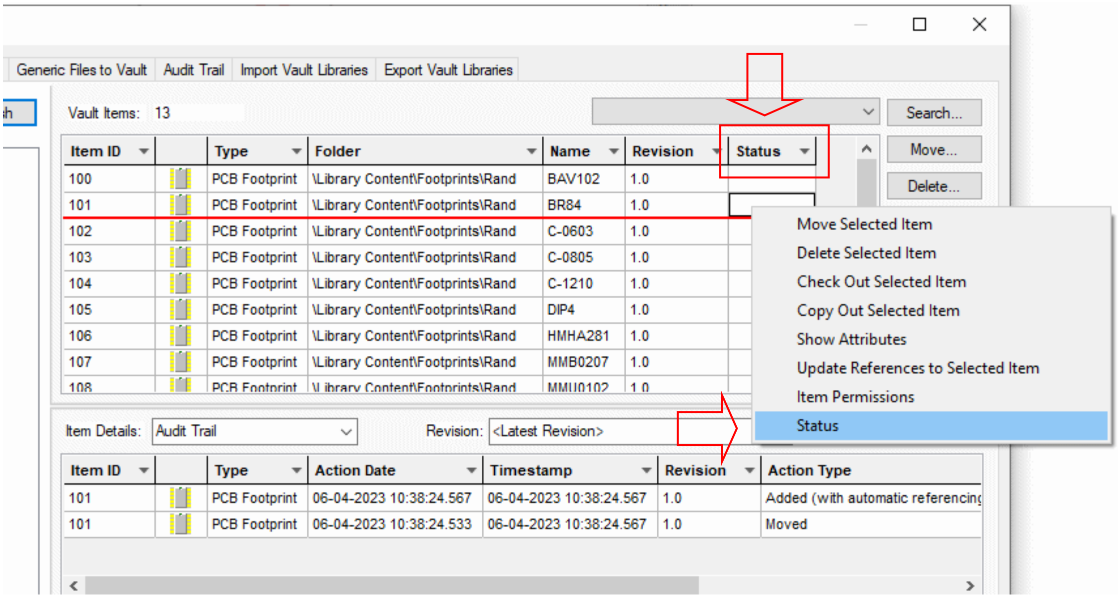
You must close and reopen the dialog in order to activate the new **Status** page.



Using this page, you can define your own status to use for the Vault. Their priorities are determined by their position in the list box.

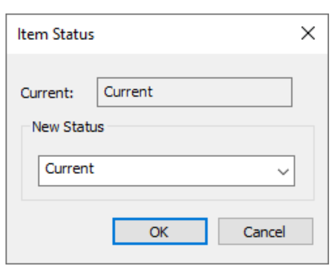


Once setup, a new column will be added to the **Vault Browser Items** grid that quickly shows you the Status of vault items.



To change the Status of an item, right click the item from the **Vault Items** grid, this will give you access to the context menu option **Status**.

Clicking this will open the **Item Status** dialog which will allow you to set the new status (from the list you've defined earlier) for the item. This new status can be used in the search mechanism.



Library Manager – Vault Operations Progress Dialog

In the **Library Manager**, a delayed progress dialog will appear when you Check-in, Check-out, Copy-out or uncheck bulk Vault items from any Vault item type (STEP Models, Parts, Symbols, etc.).

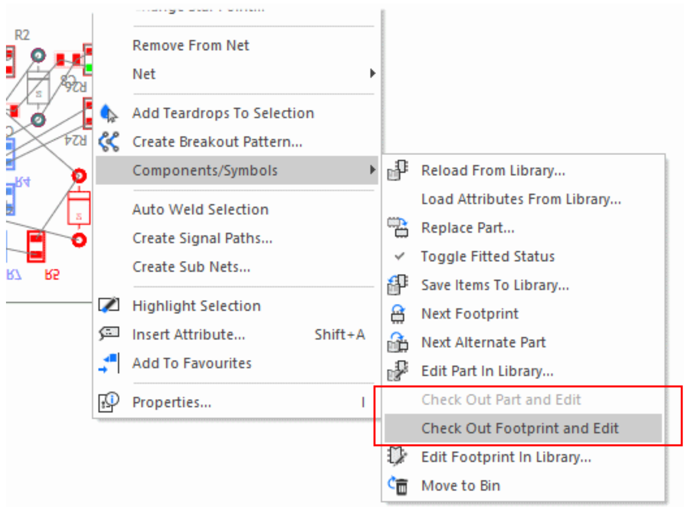
Vault Admin – Meaningful connect error messages

In the **Vault Admin Settings** page, when you attempt to connect to the **Vault database**, if an error has occurred (such as an incorrect password), an error message will appear that directs you on what your next action should be.

Check Out and Edit

When selecting a Footprint or a Part in a design that is checked in the **Vault**, there are new options on the context menu - **Check Out Part and Edit, Check Out Footprint and Edit** and **Check Out Symbol and Edit**. These allow you to check out the Part, PCB and Schematic Symbol and edit it from the Vault. The item selected must have originated from the Vault in the first place.

Once the item has been edited, it can be checked back into the Vault and then reloaded in the design.

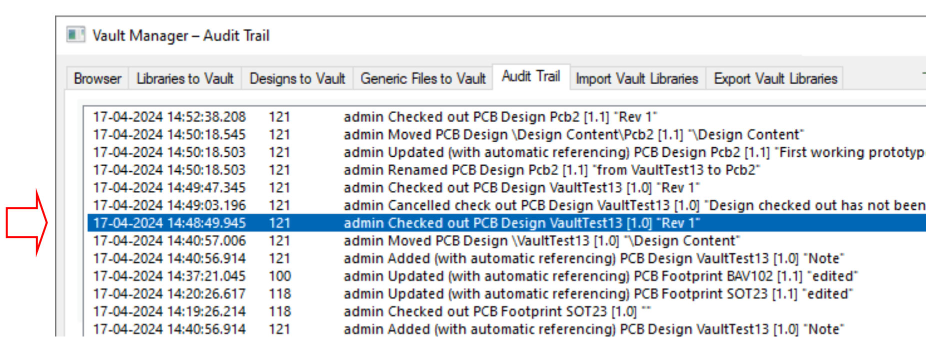


Library Export Progress Dialog

A progress dialog has been added to the **Vault Library Export** option. The progress dialog will appear whenever the export is triggered and a GUI is present (i.e. when Pulsonix is not running with -hidden command line switch).

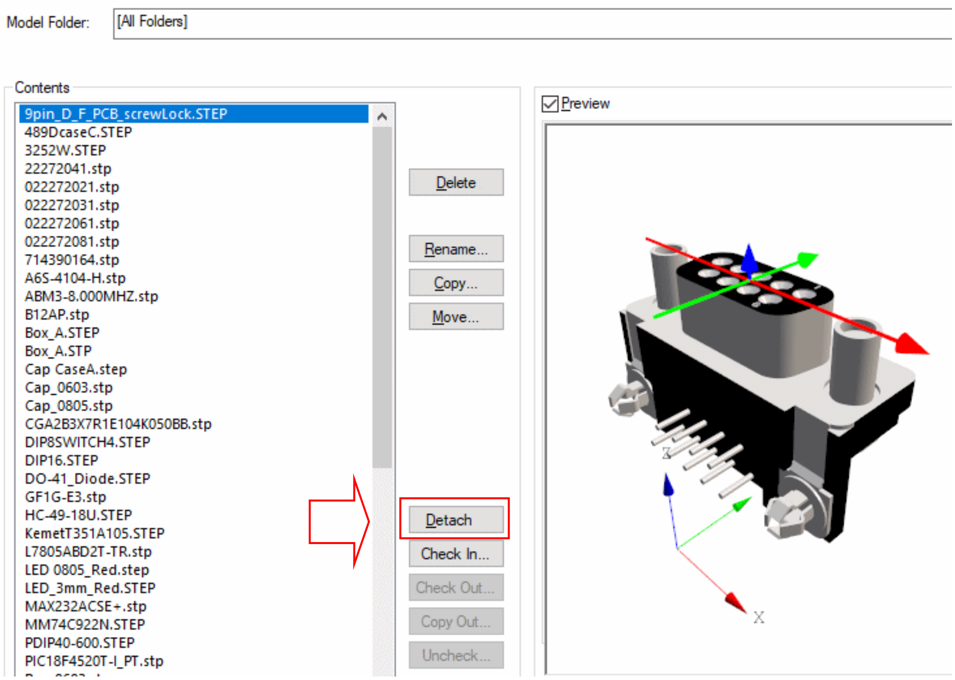
Find from Audit Trail

In the **Vault Manager**, on the **Audit Trail** page, you can double click on an item in the **Audit Trial History**, and if the item exists in the vault, it will find the item in the vault browser and jump to it. If the item doesn't exist, a new warning was added informing you that the item is no longer in the vault.



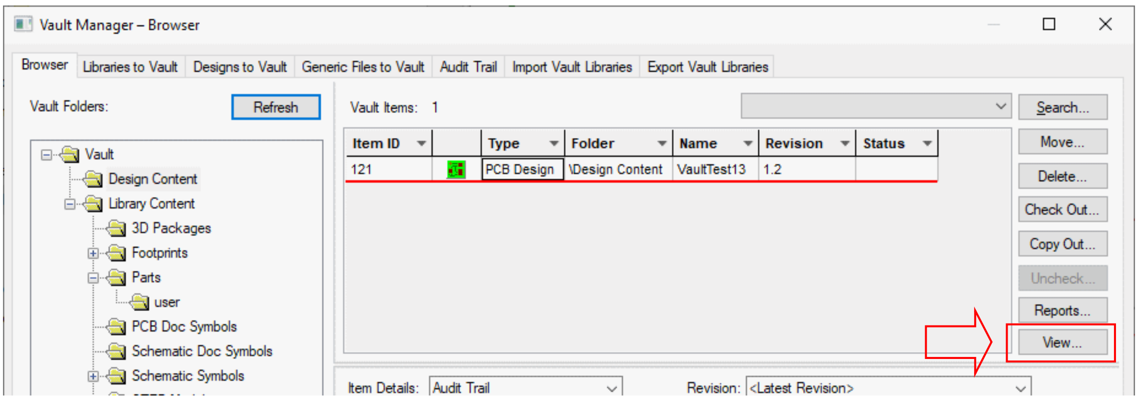
Detach Vault Items for 3D Packages and STEP Models

A **Detach** button has been added to the main **Library Manager** for **3D Packages** and **STEP Models**. You can already detach library items; this option allows you to do this to other library types.



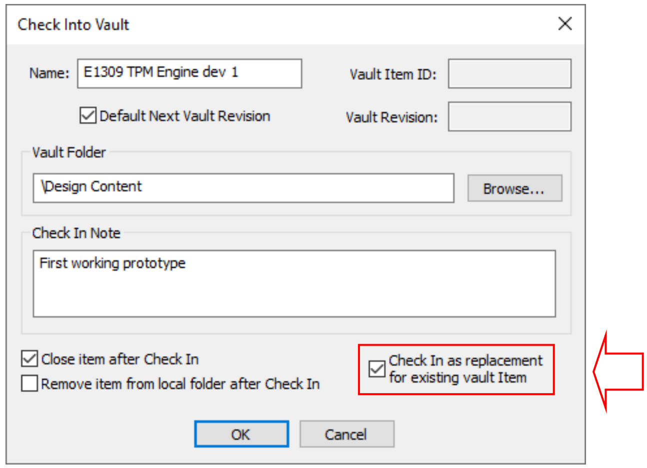
View Only Option

A new button - **View Only** has been added to the **Vault browser** for **designs** (only design items). On selection, this will display the design in the appropriate design editor but in **View Only** mode.



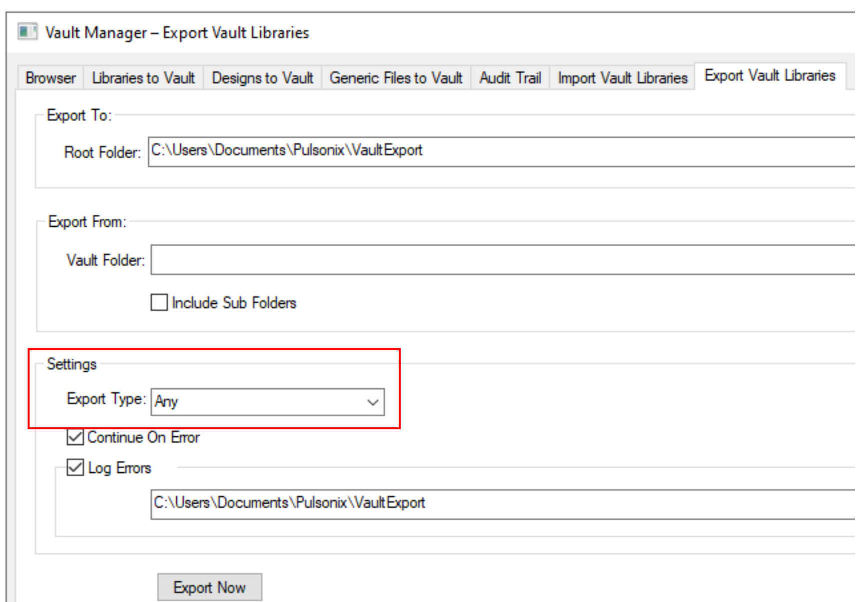
Check in as Replacement

A new check box has been added to the **Check Into Vault** dialog for **designs** - **Check in as replacement for existing vault item** option. It allows you to replace an existing vault item with a local copy as a new version. This is already available for Library items, this new option is enabled for Designs.



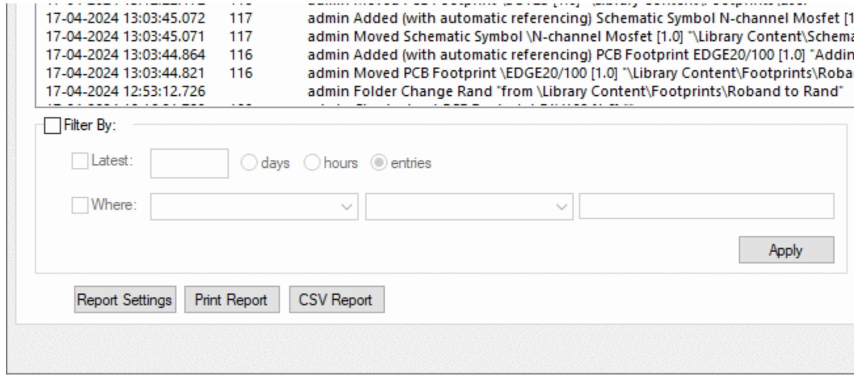
Library Export – Export by Type

You can now specify the library **type** that you wish to export from the Vault. Using the **Export Vault Libraries** page in the **Vault Manager**, you can specify the type using the drop-down box **Export Type** under settings. This gives you the option to select from all the supported library types or all of them. This new feature is also available through scripting and PLM.



Print & Save Audit Trail

The Audit Trail page in the Vault Manager has 3 new buttons - **Report Settings**, **Print Report** and **CSV Report**.

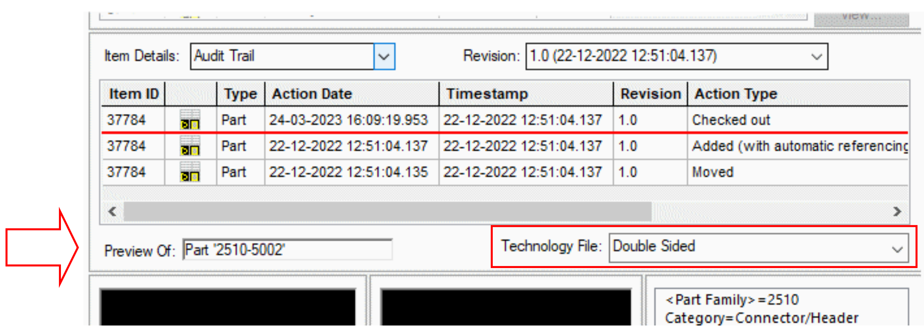


Report Settings – options to choose text setting (Shrink or Wrap). Shrink will reduce the text font to fit the page and Wrap wraps the text if it doesn't fit the page width.
Print Report – displays the **Print Setup** dialog and prints the audit trail history based on the Report Settings.
CSV Report – generates a CSV file of the audit trail history.

Technology Files in Vault

Added tech file in vault browser for symbol and footprint.

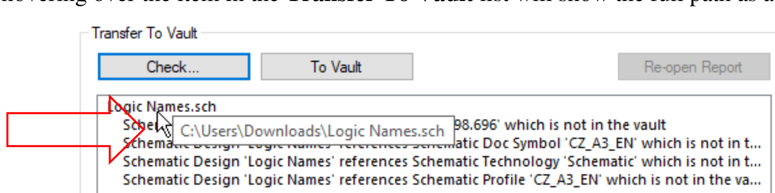
A **Technology File** can now be defined for the display in the **Preview** windows and for when editing the Footprint or Schematic Symbol in their respective editors. This works the same way as if edited from the **Library Manager**.



Transfer To Vault - Show full file path

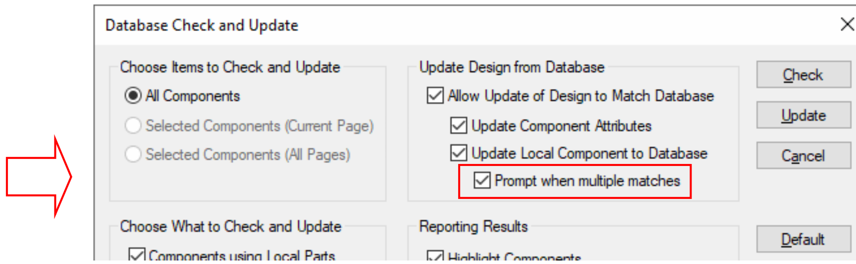
A new check box is available in **Vault Options** called **Show full file path**.

With this box unchecked, when you transfer items into the **Vault Browser** using **Designs To Vault**, hovering over the item in the **Transfer To Vault** list will show the full path as a tooltip.



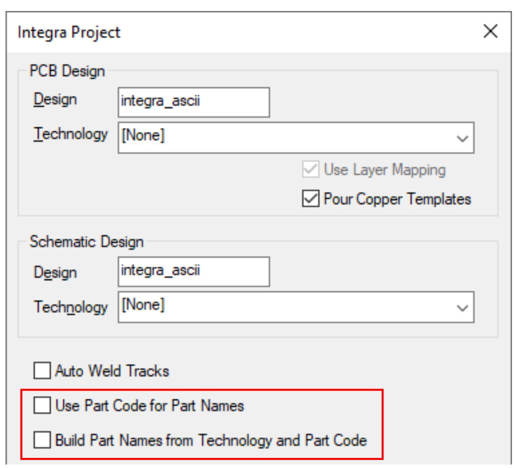
Database Connection Option (PDC) – Prompt When Multiple Matches are found

A new option has been added in the **Database Check and Update** dialog - **Prompt when multiple matches**. If multiple matches are found for a Part, a dialog is displayed showing you the record sets that matched with the Part attributes.



Import Integra Designs – Additional Selections added

For the Integra Import dialog, new switches have been added to the dialog for **Use Part Code for Part Names** and **Build Part Names from Technology and Part Code**. Previously, these were only available as registry switches.



Use Part Code for Part Names - Use this switch if the design ASCII file uses Part Codes rather than Part Names. The use of this switch will depend on how the file was created in ASCII. If the results are not what you expect, use this switch or the one below.

Build Part Names from Technology and Part Code - Use this if you wish to use a combination of the Technology attribute and Part Code attribute as an alternative to the Part Name. This is the same as the **Use Part Code for Part Names** switch but adds the Technology attribute to the name.

Scripting Changes

Scripting Attribute Visibility methods

Methods have been add for *IsAttrVisible* and *SetAttrVisible* to a *DesignItem* object in scripting. This allows you to get and set the visibility of attributes on an item in the design, in the same way as toggling the visibility check box in attribute properties.

This feature was back-fitted to V12.5.

Scripting Licenses

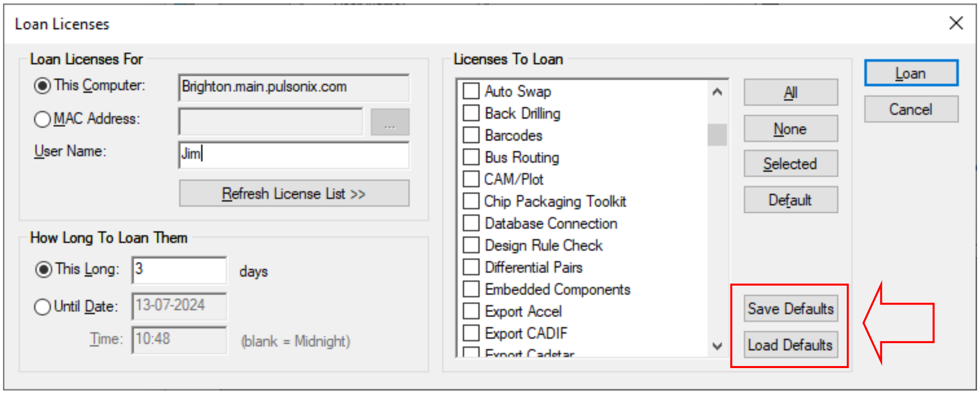
Two new licenses for scripting have been added – Scripting and Scripting Advanced Features.

The scripting license is now required to access scripting functionality (both from the scripting dialog and command line). The Scripting Advanced Features license restricts access to certain functionality available in scripting, such as accessing positional data. The functions currently blocked for this license are:

- CommonSegment – Start, Centre, End
- DesignItem – Position, X, Y
- Component – MoveTo
- TextPosition – MoveTo
- Net – AddNode, AddTrack
- Part – AddFootprintFromLibrary, AddGate, AddGateFromLibrary, Export
- PartLibrary – Import
- SymbolLibrary – Import
- Track – AddArc, AddLine

Network Licensing (NLS) – Save / Load Loan Defaults

In the **License Manager**, when loaning out a license (using the **Loan Out** button), there are two new buttons - **Save Defaults** and **Load Defaults.** These allow you to save the current list of checked licenses to be available to load in the future.



Document Verify During Save

Newly saved documents will now be verified immediately after saving, warning you if the design has become corrupt. Corrupt documents will never overwrite an existing uncorrupted document and will be saved to a new name.

In addition, backups won't be created if a saved document is corrupt.

A variant of this feature was back-fitted to V12.5 to add protection. The option in Version 13 is significantly faster on saving.

PLM Interface Changes

If you have the optional PLM license, the PLM interface *WriteReport* command now has an additional parameter, *View,* to display the generated report on screen.

This feature was back-fitted to V12.5.