Pulsonix Design System

V11.0 Update Notes

Copyright Notice

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 11/05/21 iss 4

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 11.0 Update Supplement

Installing the New Version of Pulsonix

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

To install Pulsonix, double-click on the download executable and wait for a short time. You may have been supplied a CD, in which case insert it and wait for it to run. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 11.0 over your existing V10.5 installation. If upgrading from V10.5, you can install it alongside the older version if you prefer. In any case, you do not need to uninstall the old version first unless you wish to remove it from your hard drive.

Licensing

Version 11.0 requires a new license if you are a new user or upgrading from any older version of Pulsonix earlier than V10.0or V10.5. 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 11.0 Update Notes

These Version 11.0 Update Notes are available in the Pulsonix product installation folder under \documents. Alternatively, select the direct link within the product itself: **Help** menu **| Online Manuals> Pulsonix V11.0 Updates Notes**. You can also find them on the Pulsonix web site.

The current and all previous Update Notes are available on the Pulsonix web site under **Documentation**.

Installation

The installation has been changed so that all Pulsonix ‘documents’ (Master Libraries, Technology files etc.) are now located under user\documents\Pulsonix11.0 rather than being placed in public documents\PulsonixXX

New In Version 11.0

Report Maker – List of Files

When creating a report using **Report Maker**, a new list command option **List of Files** has been added which when the report is run, will list all the files within a chosen folder.

The destination can be edited by double clicking the command in the format script section. A folder path can be added by browsing a folder using the browse button, or being picked up by a report variable or by typing in a path (this benefits from relative paths, so folders within the design folder can be selected by just typing in that folder's name). Whilst within the List of Files indentation in format script, another new command **File Name** is available which reports the name of the file that is found.

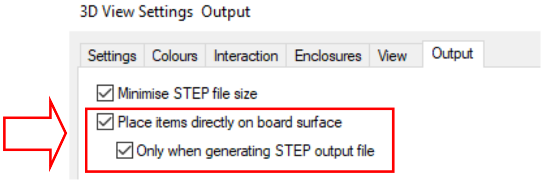
Part Edit – Force Display of user attributes

When editing a Part in the **Part Editor**, the **Force Attribute Displayed** dialog, available from the **Edit** menu, now includes two lists for you to define what attributes you want to force displayed when adding the Part to a schematic or PCB design. If not already in the symbol, the attribute positions will be added underneath each other in the order from the list.

3D View Settings (STEP) Change

To avoid graphical issues (dithering, also known as Z-fighting) when drawing two or more items at the precise same location, components are added to the STEP data using a tiny Z-offset to render them very slightly above the surface of the board. The same is true of additional design data that might be enabled (such as Pads).

If for some reason you need this tiny offset to be removed so that items are rendered at precisely the same Z value then two new check boxes on the **Output** tab of the **3D Settings** dialog allow you to do this:



The default setting for **Place items** check box is unchecked. On selection of this box, the next check box becomes available. This second check box, **Only when generating STEP output file**, controls whether the decision to place items directly on the board is done in the on-screen Preview or only when writing data to a STEP file. Note that if you check this box there is a performance overhead as the 3D data has to be regenerated before being written to the STEP file so that items are positioned as required in the correct position on the Z axis.

Customize – Fixed Icons

The default icons for the **SVG** and **STEP** commands on the **Output** menu bar were incorrect and have been updated to use the correct icon to represent the command. To view this change, you will have to reset the toolbar in the **Customise** dialog in order to get the fixed icons.

New Direct-X Graphics Engine

Pulsonix 11 now supports Direct-X graphics. From the **Options** dialog, **Display** page, there is a new slider for **Hardware Acceleration** to enable the new graphics engine. For large designs, this may take a moment to rebuild the graphics.

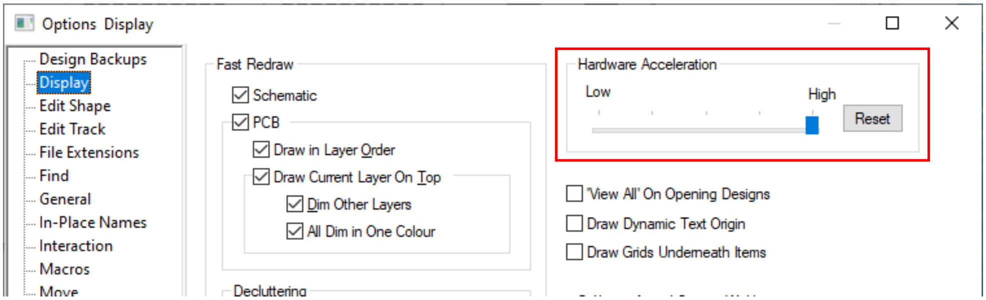
This slider will allow you to choose the level of hardware acceleration you wish to be applied when rendering graphics in a design window and balance the competing needs of initial graphics generation time, pan and zoom performance and amount of memory used.

The slider provides 5 possible settings; the first, leftmost position, indicates minimal hardware acceleration with purely GDI drawing and is the equivalent of the old **Enhanced Graphics** setting being unchecked. The **Enhanced Graphics** button has been removed from the dialog.

The second position is the equivalent of **Enhanced Graphics** being enabled and, like in 10.5, provides anti-aliased circle drawing but no performance improvement.

Positions three to five provide new Version 11.0 settings and indicate varying degrees of hardware acceleration utilising the graphics card GPU. The higher the position, the more improved the pan and zoom performance will be, but at the cost of a greater initial graphics generation time and increased memory usage. Moving the slider to the **High** position will utilise all the new graphics features and use any memory available and required.

The **Flicker Free Redraw** option has also been moved from the **Display** page.



International Language and Extended Character Support

Pulsonix version 11 has been switched to use the full Unicode instruction set. This means full international language and extended character support. This will support special characters like Ω, ≠, ≤, ≥, ±, ∅ for example and also Chinese and Korean languages. Support is through all design editors, libraries, import filters and exports such as Gerber, Netlists, Parts Lists and all 3rd party products.

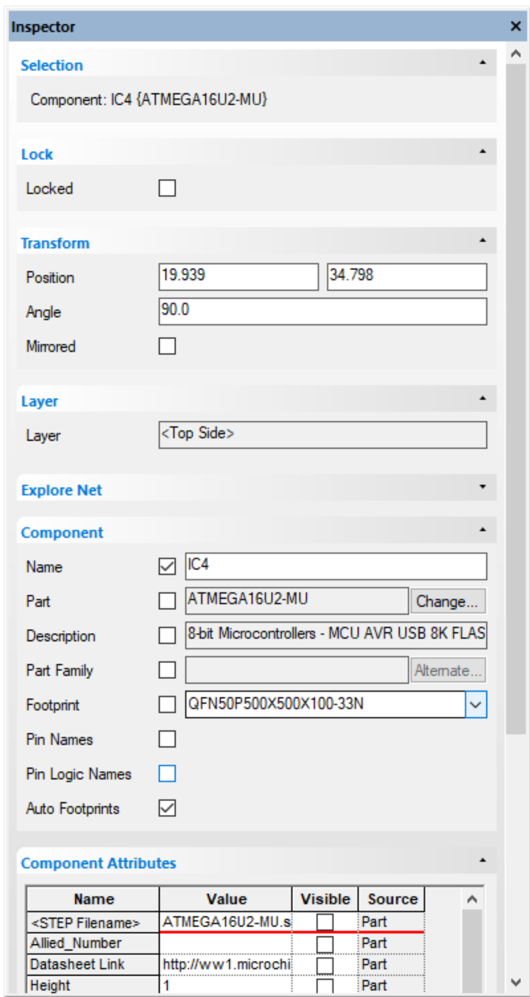
Inspector Bar (Dynamic Properties)

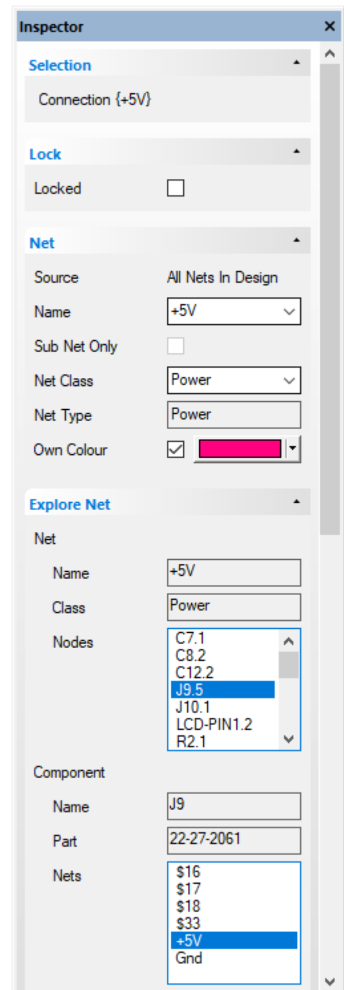
The new **Inspector Bar** acts as a dynamic **Properties** windows, however, it is different.

Accessed by right clicking on the Pulsonix framework and selecting **Inspector** off the list of bars available, or access it from the **View** menu and selecting **Inspector Bar**.

Generally speaking, the Inspector Bar would be left either docked open or floating so that it is accessible instantly.

With an item selected, the Inspector Bar will display Properties for that item, for example, a selected Component in PCB:





Context Menu Options

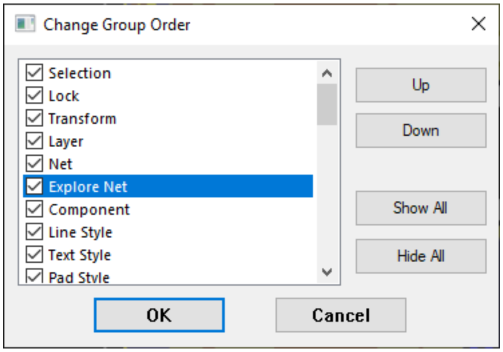
By right clicking on the Inspector Bar, you can access additional commands:

Change Group Order

Select **Change Group Order** to rearrange the ordering of the 'groups' or sets of properties displayed in the Inspector Bar.

Use the **Up** and **Down** arrows to move the selected group up or down in the Inspector Bar list.

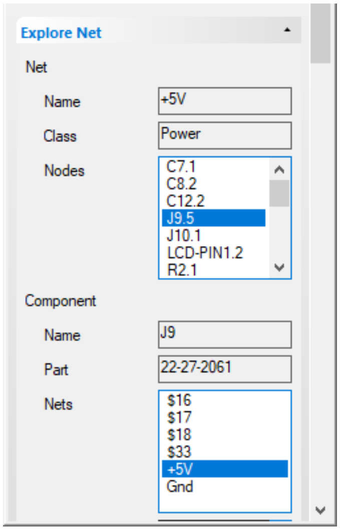
Groups can be shown or hidden in the Inspector Bar using the check boxes next to the group or by using **Show All/Hide All**.



Explore Net

The group **Explore Net** enables you to see the nodes on a selected net and the nets on the component (if the node is from a component). This functionality also works in reverse, so for a selected component, the Inspector Bar will show all nets on that component.

Selecting a node or net from one of the lists in the group will update the other list (Net or Component or vice-versa), allowing the connected nets and nodes to be traversed (or explored).

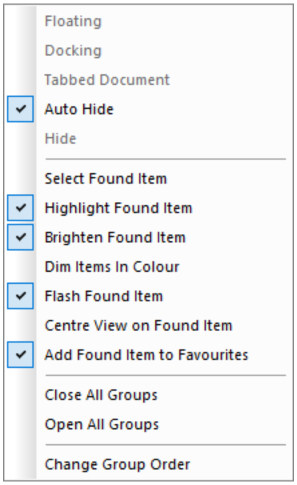


Single Clicking to Navigate Nets & Nodes

Single click will 'navigate' net and nodes, using a double click will highlight net.

Find Options

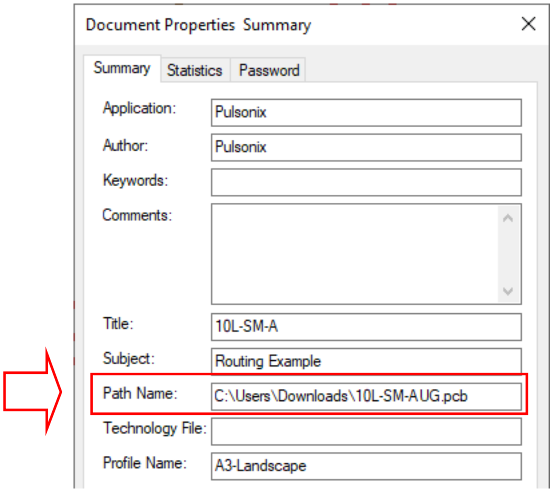
This offers the same finder functionality as the **Find Bar**, with the Find Bar options being used here as well. These options can be viewed and changed from the Inspector Bar by right-clicking anywhere on the bar and hovering over the **Find Options** context menu (see below).



Changes to Document Properties Dialog

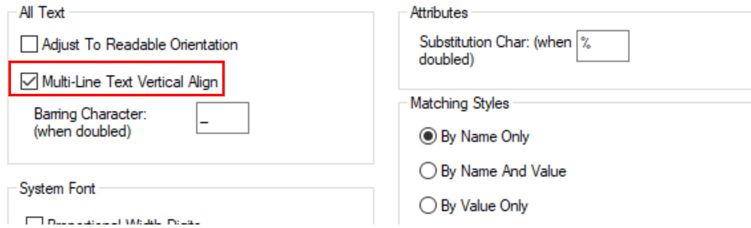
Change to Summary Page

A new item on the **Summary** page of the **Design Properties** dialog has been added. This shows you the current full path name of the design file.



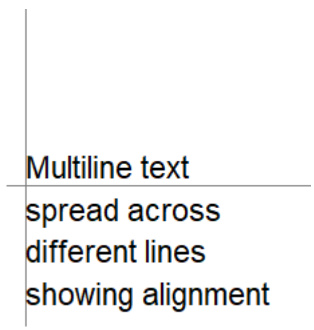
Vertical Text Alignment for Multi-line Text

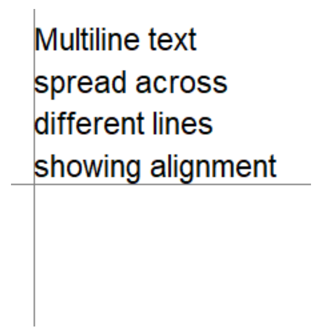
There is a new switch in **Design Settings, General,** under **All Text** to set **Multi-Line Text Vertical Align**. When selected, this causes all multi-line text to vertical align using the height of all the text, not just the first line. It will change the vertical alignment of all multi-line text in your design so care must be taken if applying this after already having added multi-line text, especially on electrical layers.



The two images below show the effect of this switch. To clarify where the alignment is taken from, horizontal and vertical construction lines have been added to the text to show the movement.

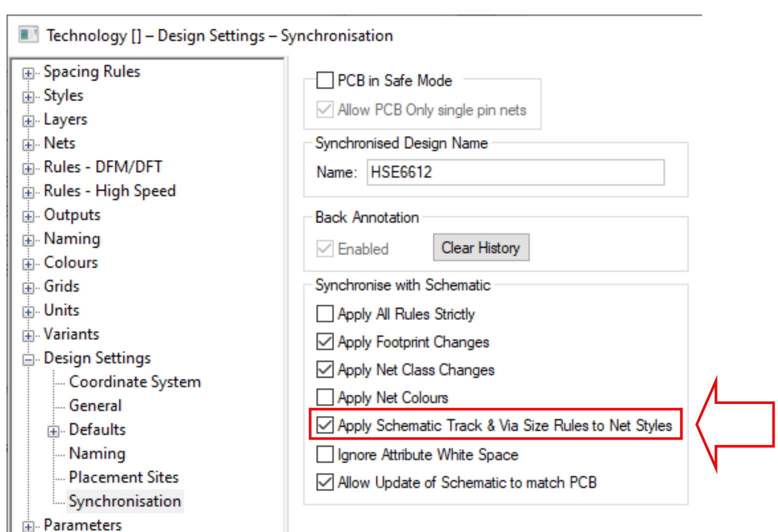
The left image is before switch is activated (and as current product), the right-hand image is after:





Option for Track & Vias styles defined in SCM to be Translated to PCB

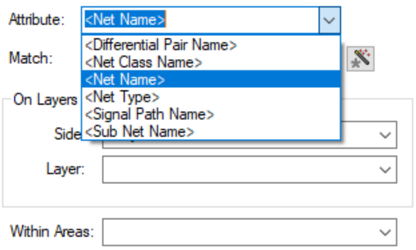
A new option in the **Design Settings** option under **Synchronisation** now allows **Net Styles** in PCB to be created from the **Track & Via Size Rules** in the Schematic. Use the **Apply Schematic Track & Via Size Rules to Net Styles** switch to enable this feature.



This means, where you need to specifically define Track and/or Via Styles in the Schematic, they can now be passed through into your PCB where they will be used in Net Styles. Both the **Translate To PCB** and **Synchronise Designs** options will use this feature once the switch is set.

If **Translating to PCB** to create the initial design, your PCB Technology used must already have this switch set in order to pass across the settings. If it isn't set, then once set, use the **Synchronise Design** option to update the design with these settings.

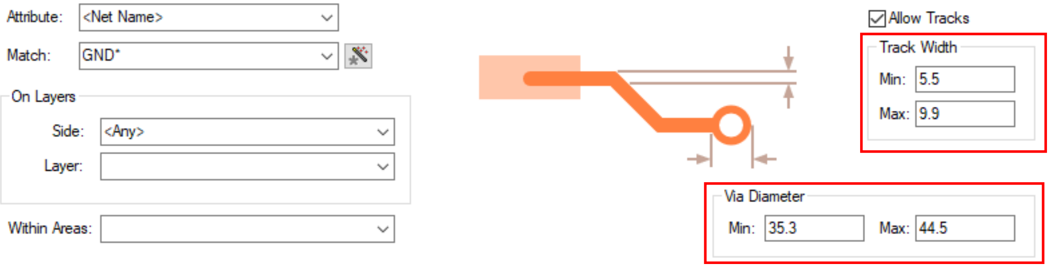
The **Track & Via Size Rules** in the Schematic should be defined in accordance to your requirement. These can be generic such as all **Net Names** in the design, or more refined such as specifically named nets such as DQS*. Other parameters can be selected from the drop-down list as well as **Layers**, **Sides** and **Within Areas**.



Design Settings Synchronisation

Once the **Apply Schematic Track & Via Size Rules to Net Styles** switch has been checked in the **PCB Design Settings Synchronise** dialog, you can then define the **Track and Via** styles in your **Schematic** design.

From in the **Schematic** design, use the **Technology** dialog and **DFM/DFT Rules – Track & Via Size Limit** dialog to add rules that will be passed into your PCB design.

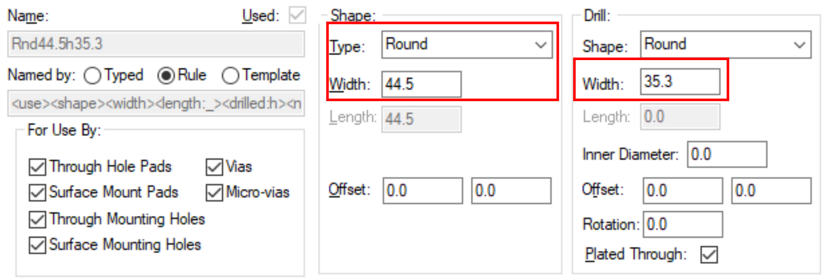


Translate to PCB

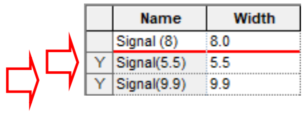
During **Translate to PCB**, with this switch set in your **PCB Technology file**, this setting will be initially applied to the new design and your **Schematic Track & Via Size Limit Rules** will be defined in the PCB

If the **Track & Via Size Limit Rules** are applied to the PCB, it will add new **Track** styles and a new **Pad Style** to the **Technology**. Two new Track Styles will be created based on the Min and Max Track width defined. The Pad style created will use the Min and Max Via Dimeter to create a pad of the Max diameter size with a drill using the Min Diameter value.

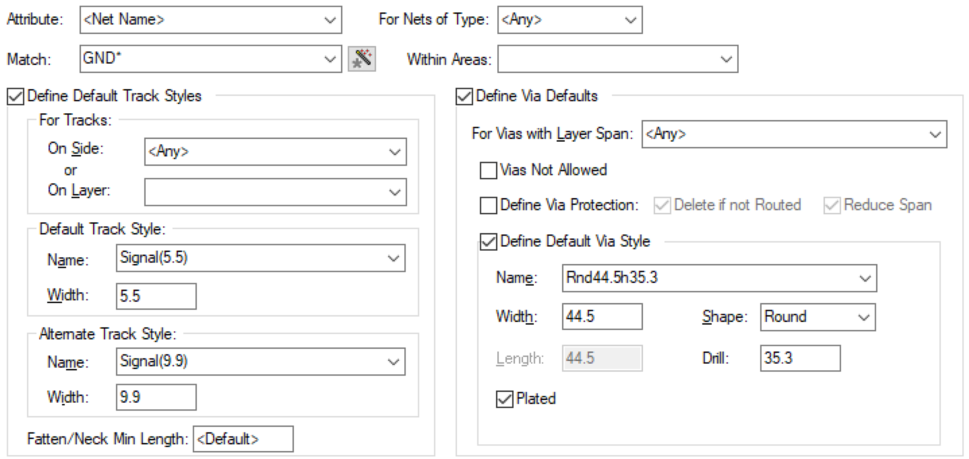
A new **Via Pad** style is added to the **Pad Styles** dialog:



New **Track** styles are added to the **Track Styles** dialog:



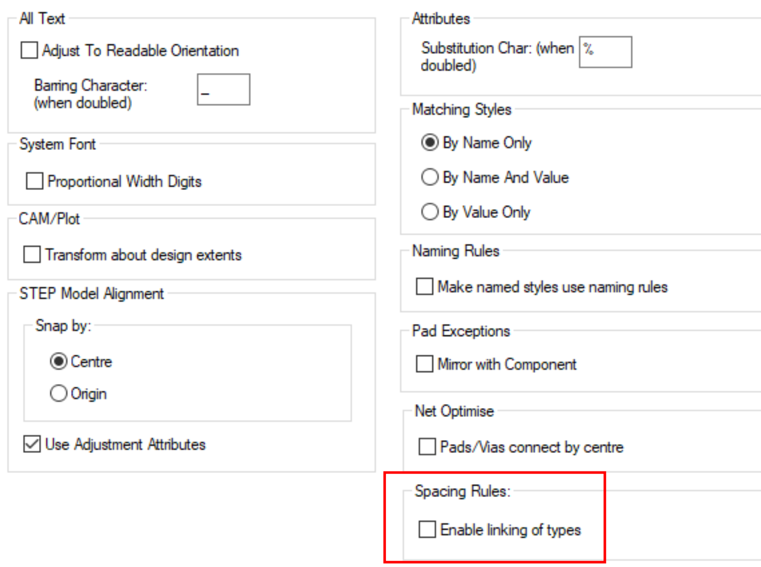
Matching settings in the **Net Styles** dialog for the nets requiring them will also be created using the new (or matched) Track and Via Styles.



Options Dialog Changes

Option for Spacing Rule Grids

A new option, **Spacing Rules - Enable linking of types** has been added to **Design Settings, General**. When this is checked, some rows and columns on the Spacing grids will disappear (provided the spacings are the same), and the values will be linked together. This setting will be saved with your design so it is retained.

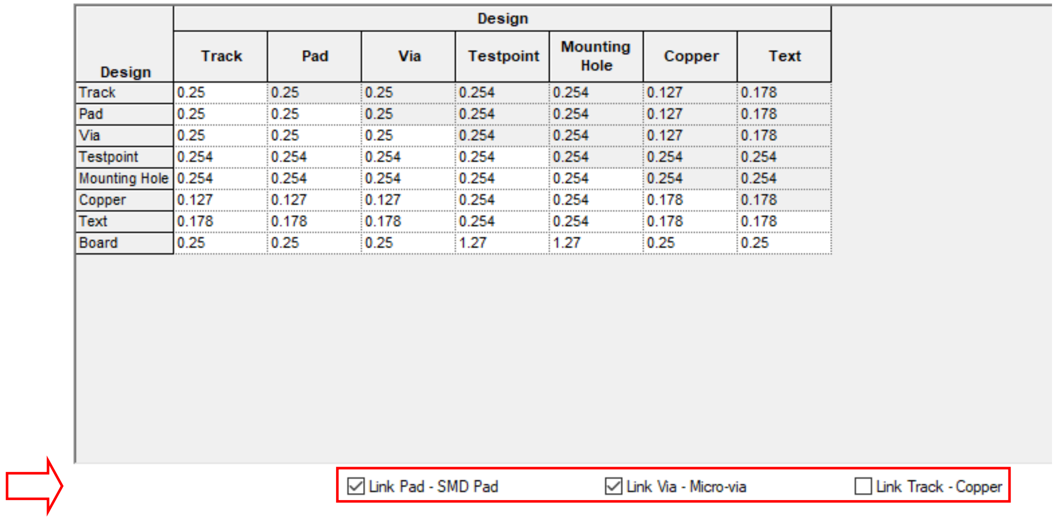


You can show the hidden values by unchecking the appropriate options which appear under the spacing grid.

Before linking, the Spacing grid looks like this:

	Design								
Design	Track	Pad	SMD Pad	Via	Micro-via	Testpoint	Mounting Hole	Copper	Text
Track	0.25	0.25	0.25	0.25	0.25	0.254	0.254	0.127	0.178
Pad	0.25	0.25	0.25	0.25	0.25	0.254	0.254	0.127	0.178
SMD Pad	0.25	0.25	0.25	0.25	0.25	0.254	0.254	0.127	0.178
Via	0.25	0.25	0.25	0.25	0.25	0.254	0.254	0.127	0.178
Micro-via	0.25	0.25	0.25	0.25	0.25	0.254	0.254	0.127	0.178
Testpoint	0.254	0.254	0.254	0.254	0.254	0.254	0.254	0.254	0.254
Mounting Hole	0.254	0.254	0.254	0.254	0.254	0.254	0.254	0.254	0.254
Copper	0.127	0.127	0.127	0.127	0.127	0.254	0.254	0.178	0.178
Text	0.178	0.178	0.178	0.178	0.178	0.254	0.254	0.178	0.178
Board	0.25	0.25	0.25	0.25	0.25	1.27	1.27	0.25	0.25

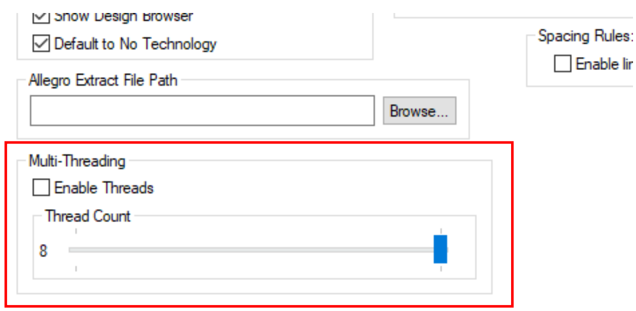
With **Enable linking of types** checked, the Spacing grid now looks like this:



Design	Track	Pad	Via	Testpoint	Mounting Hole	Copper	Text
Track	0.25	0.25	0.25	0.254	0.254	0.127	0.178
Pad	0.25	0.25	0.25	0.254	0.254	0.127	0.178
Via	0.25	0.25	0.25	0.254	0.254	0.127	0.178
Testpoint	0.254	0.254	0.254	0.254	0.254	0.254	0.254
Mounting Hole	0.254	0.254	0.254	0.254	0.254	0.254	0.254
Copper	0.127	0.127	0.127	0.254	0.254	0.178	0.178
Text	0.178	0.178	0.178	0.254	0.254	0.178	0.178
Board	0.25	0.25	0.25	1.27	1.27	0.25	0.25

Thread Count Definition

A new option, **Thread Count** has been added to **Options, General**. The maximum value of the slider is automatically presented based on the number of logical processors your PC has.

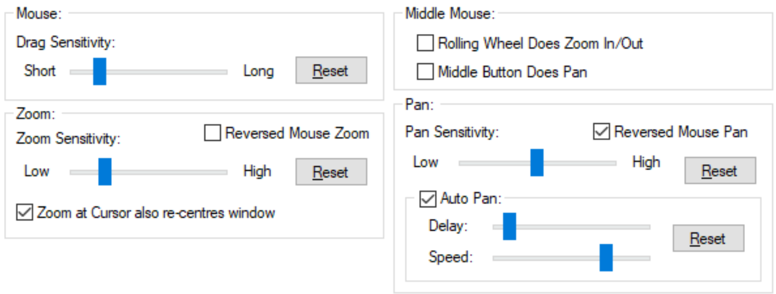


If **Enable Threads** is checked, multiple threads will be used to perform task that utilise it, including but not exclusive to: CAM Plot Gerber Export, Copper pour, Design Rule Checks and Filling the Rules spreadsheet.

The **Thread Count** slider defines the number of threads used when performing the above tasks.

New Options page for Pan & Zoom functionality

The **Pan** and **Zoom** settings in **Options**, **Interaction** had been moved to a new page **Options**, **Pan & Zoom**. This page has been added to declutter the **Interaction** page and add new functionality (namely, **Auto Pan**, see below).



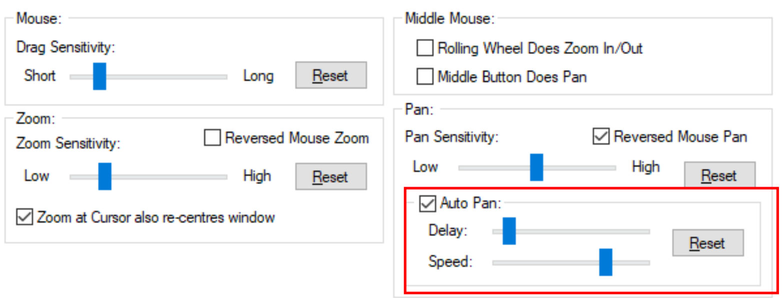
New Auto-Pan feature

Auto-Pan has been added to Pulsonix. It has been added to the **Options**, **Pan & Zoom** page. This allows you to automatically scroll (pan) your design by moving the mouse to any edge of the design area (the target region). When the mouse gets to within 12.7mm (1/2 inch) from the edge of the design area, Auto-Pan is activated.

The **Delay** and **Speed** can be defined for this option.

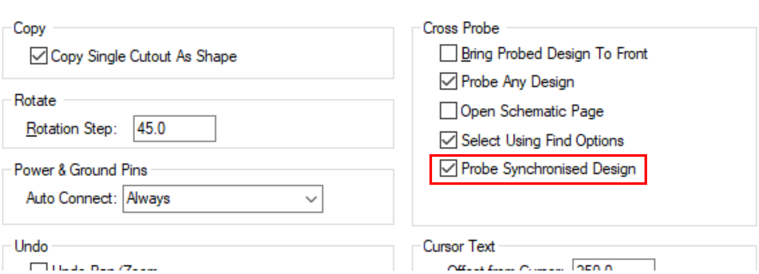
> **Delay** is the time it takes for the Auto-Pan option to 'react' when the cursor is moved into the target region.
>
> **Speed** is the speed of the pan once Auto-Pan is activated. If the slider is to the left most side, the speed will be slow, to the right side, it will be much faster.

The **Reset** button is used to reset the Delay and Speed sliders back to their default factory settings (these have been predefined to be good general working settings).

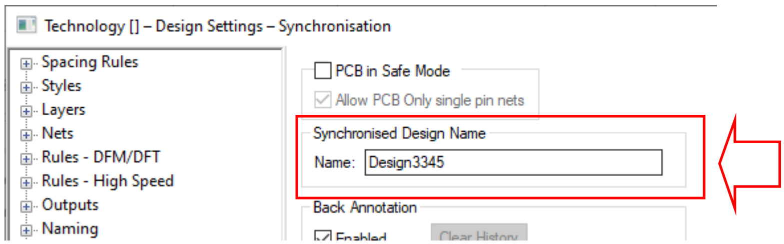


Probe Synchronised Design option added to Cross-Probe

A new option, **Probe Synchronised Design** has been added to the **Options** dialog, **Interaction** page under **Cross Probe**.

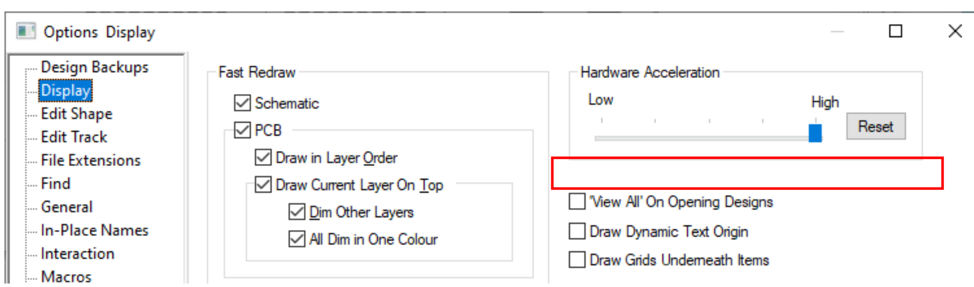


Selecting this option enables cross probing to take into account synchronised designs which are assigned in the **Design Settings Synchronisation** dialog. When selected, cross probe will check if the nominated synchronised document exists, and if so, selects the probed component in that design. If the synchronised file is not found or this option is disabled, then cross probing will run as usual.



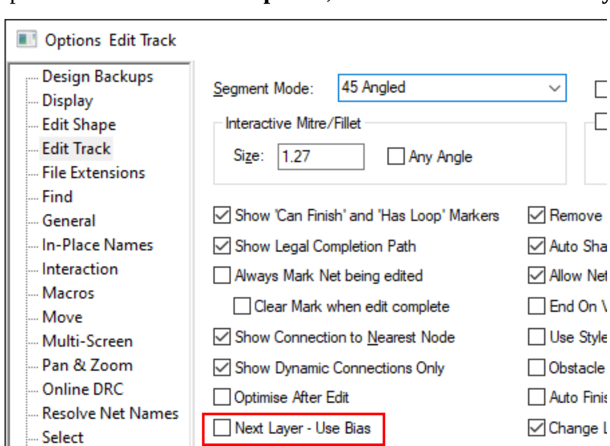
Removal of Fast Locate Option

The **Fast Locate** option on the **Options** dialog and **Display** page has been removed (shown below as the empty red box). This setting was used to enable and disable the enhanced picking but is no longer required. It will always be set on now by default but unavailable for changing.



Edit Track Option for using Layer Bias

A new option has been added to **Options**, **Edit Track** called **Next Layer – Use Bias**



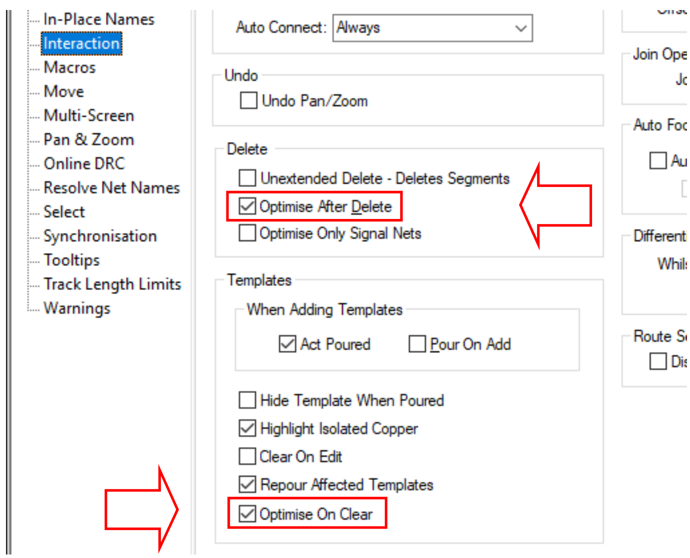
When editing a track, you now have the option to utilise the **Layer Bias** defined in your **Technology Layers**. This means when using the **Next Layer** and **Previous Layer** commands, it will skip layers with a bias of **Power Plane**, **No Tracks** or **Minimum Tracks**.

With this option left unchecked, the Next/Previous Layer commands will use all electrical layers of any bias.

Optimise On Clear Template Option

On the **Options**, **Interaction** dialog, **Optimise On Clear** has been split out from the **Optimise On Delete** option to make it clearer that an optimise will be performed when you clear a template.

There is a new **Optimise On Clear** check box under the **Templates** section to enable/disable this.



Clear All Templates – Optimise now Multi-thread

Multiple threads can now be use when optimising nets after clearing all templates. This significantly speeds up the processing time for designs with large nets.

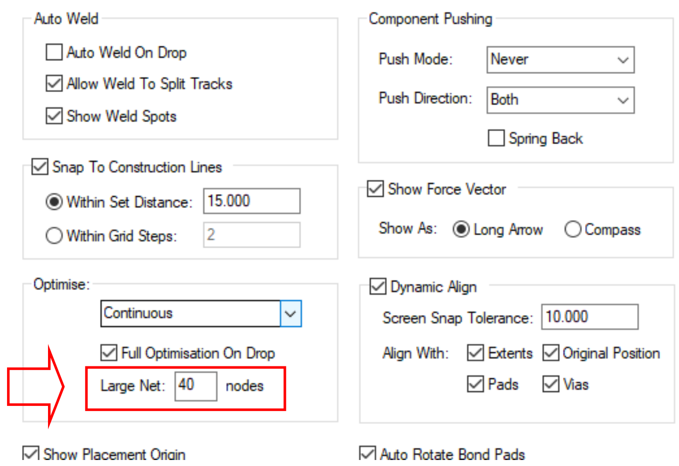
The selection to **Enable Threads** for **Copper Pour** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Optimise Settings for Large Nets

To reduce the impact of large nets on how **Optimise** runs, there is a new **Large Net** setting on the **Move** page of the **Options** dialog. This is additional to the setting that controls how optimise works on placing components (Continuous, Never, On Drop etc.).

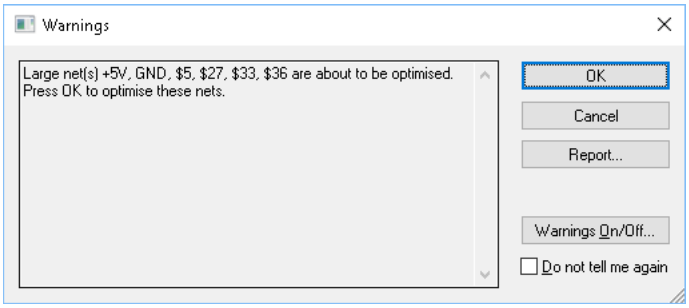
This new option instructs Pulsonix to skip nets that have more than the specified number of nodes because they are 'expensive' to optimise. The main **Optimise Nets** command will still optimise all nets regardless of complexity, this change will improve responsiveness during interactive editing for large nets.

Note: a node is any connection point - pad, via, star point, test point, junction or copper.



Large Net Warning on Optimise All Nets

An additional **Optimising large net** warning has been added when optimising all nets. The warning is shown at the very beginning when **Optimise** is run. It lists all large nets that will be optimised. This gives you the option to not optimise these large nets.



A large net is identified in **Options, Move** under the **Large Net** setting (see above).

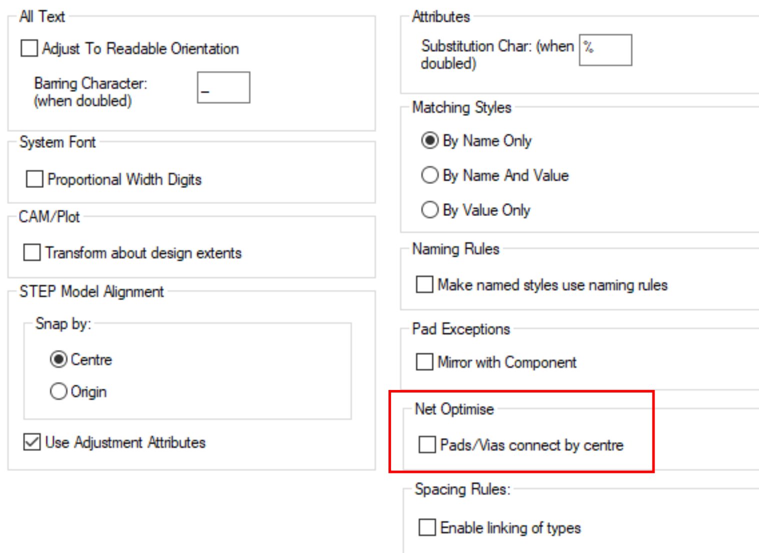
Optimise All Nets Multi-Threading

The ability to utilise multi-threading when **Optimising All Nets** has been added. This change speeds up the overall processing time of this function.

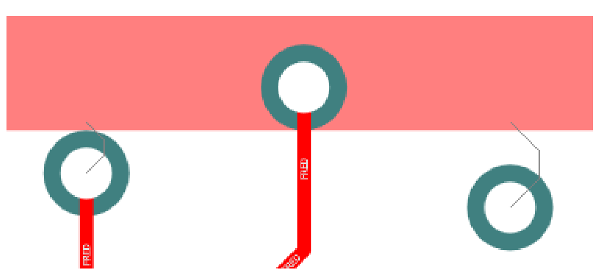
The selection to **Enable Threads** for use during **Optimise All** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Net Optimise for Pad Centres within a Shape

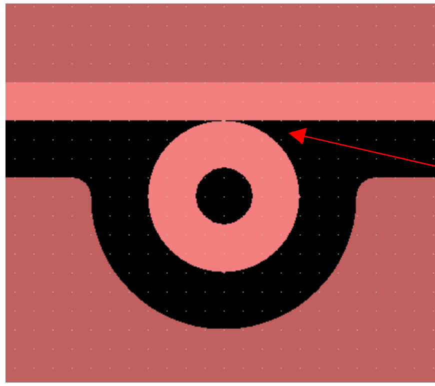
There is a new option in **Design Settings**, **General,** called **Net Optimise.** This forces a connection (and therefore it will not be properly connected) unless the centre of a pad or via is within the shape of the other item. This prevents pads which just touch the edge of a shape from being considered connected and thus avoid thin bridges which may or may not be connected after manufacture. This setting is saved with the design so that it isn't lost.



In the example below, three clear scenarios are shown where this option is enabled and **Optimise** has been run; one where the pad is touching and a connection exists, the middle one where the pad is completely enclosed and no connection is required, and the third pad where it is clear of the shape and shows a connection.



When **DRC** is run, this forms part of the **Net Connectivity** check (which checks for unrouted connections).

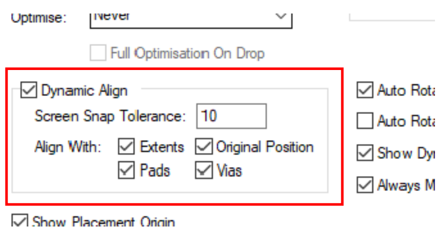


The pad touching here is considered a ‘thin bridge’ which may cause manufacturing issues.

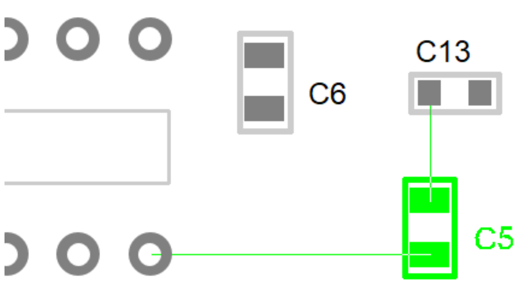
Dynamic Align of items

There is now an option to dynamically align items being moved to static items in a Schematic or PCB design. From the **Options** dialog, **Move** page, there are options to set for refinement.

This option is compatible with; Components, Symbols, Pads, Vias and Text.



When an item is moved, with this option enabled, ‘alignment’ lines are displayed:



Screen Snap Tolerance – This is the value for the tolerance at which an item will snap to another item during move. The tolerance will remain the same screen distance regardless of the current zoom level. This value will be set in the current design units.

Align With

Extents - Any item will snap to another item's extents during move. Extents is the total circumference of a shape.

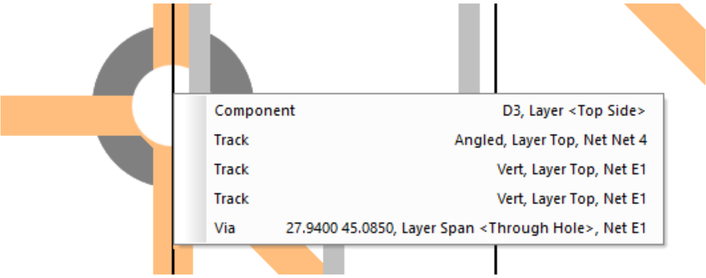
Original Position - Any item will snap to its Original Position during move.

Pads - Pads, Components/Symbols and Vias will snap to another item's Pad during move.

Vias - Pads, Components/Symbols and Vias will snap to Vias during move.

Select from popup list

Holding down the **Alt** key and **selecting overlapping items**, a context menu will list all items at that position, allowing you to select a specific item based on its description rather than the item. This is an alternative to selecting the first item, then pressing the <N> key to get the new selected item within the picking tolerance.



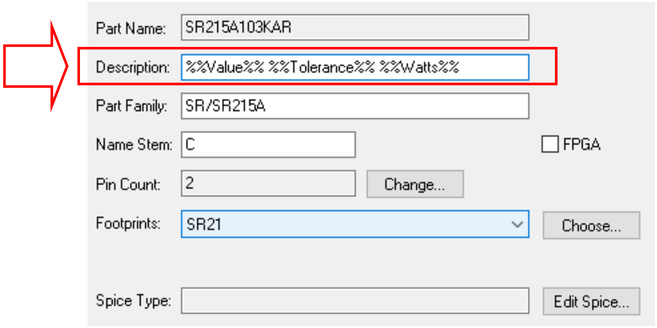
This feature can be disabled in the **Select** section of the **Options** dialog and **Select** page.

Using this option within a mode

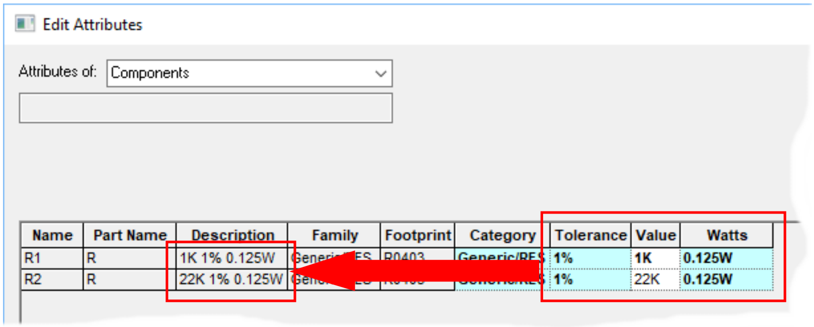
As well as using this popup a select mode within the design, you can also use it within other modes to obtain the correct item for selection. For example, use it within the **Insert Dimension** or **Measure** tools.

Part Editor - Dynamic Attributes for Part Description field

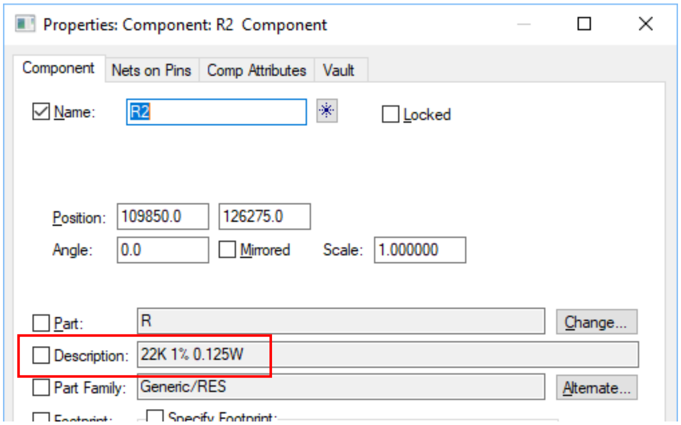
For a **Part** in the **Part Editor**, the **Part Description** field can be created using an Attribute substitution. For example, the Part Description may be made up like this: %%Value%% %%Tolerance%% %%Watts%%



Once the Component is added to the **design** and the **Attribute Values** are satisfied, the Part **Description** field will be populated and viewed or reported using the relevant options, for example, **Report Maker**, **Properties**, **Attribute Editor** etc. It is viewed as a populated state in the design and can only be edited from within the Part Editor.



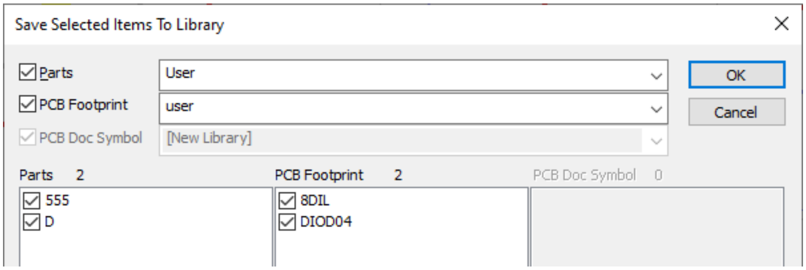
In **Properties**, the Part **Description** gets resolved like this to show the populated values:



Save Items to Library dialog changes

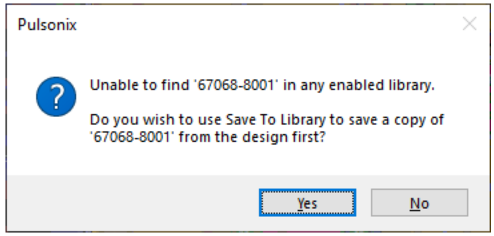
On the existing **Save Items to Library** dialog (available for a selected component in the design or by dropping a design onto the Library Manager), the list boxes have been changed to be **check list boxes**. This allows for you to specify which items should be added to the chosen libraries.

Additional **Select All** and **Deselect All** buttons have also been added to aid the selection of all items.

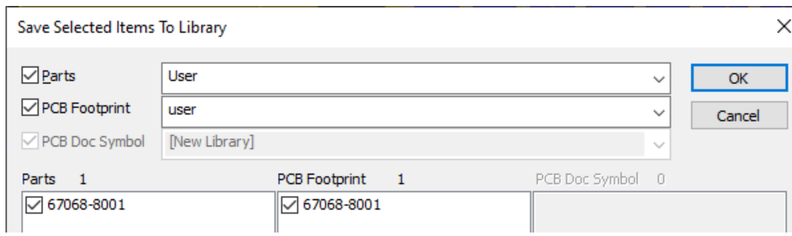


Save Missing Component Items to Library

From within a design (Schematic or PCB), when using the **Edit Part**, **Edit Footprint** or **Edit Symbol in Library** options, if the selected Component's Part definition, Footprint or Symbol does not currently exist in any library, you will have the option to use the **Save Items To Library** dialog to save it to a library first and then edit it once it's been saved.



The individual library item is displayed in the **Save Items To Library** dialog:

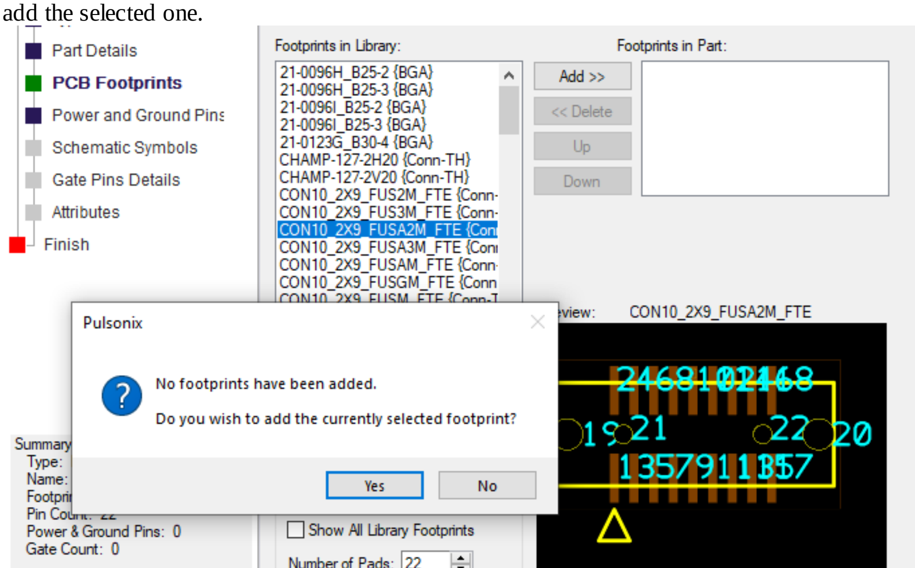


Part Wizard changes

Add Selected Footprint/Symbol

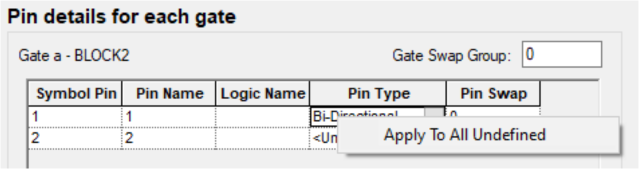
With the **New Part Wizard**, when clicking **Next'** on the **PCB Footprints** or **Schematic Symbols** page, if no Footprints/Gates have been added but one is selected in the **Footprints/Symbols in Library** list, a message is displayed giving you the option to add the one currently selected.

The message is also displayed on the symbol page if you haven't added enough gates, allowing you to add the selected one.



Pin Type cell new option to Apply to Undefined

On the **Power and Ground Pins** and **Gate Pins Details** pages, once details have been defined, you can now right click on the **Pin Type** cell and use the context menu to apply that pin type to all undefined pin types using the **Apply To All Undefined** option.



Vault Library Export to Specific Folder

There is an enhanced to allow a specific **Vault folder** to be specified as an alternative to always exporting all library data.

Use **Export From: Vault Folder** to specify a particular Vault folder from which to export library data, or leave it blank to export ALL library data contained in the Vault.

The **Browse** button will display a dialog to allow you to browse the Vault folders to find the appropriate one.

Check **Include Sub Folders** if you wish library data to be exported from the selected folder and all of the sub folders it contains.

The **VaultExport** command is available for a Pulsonix Command File.

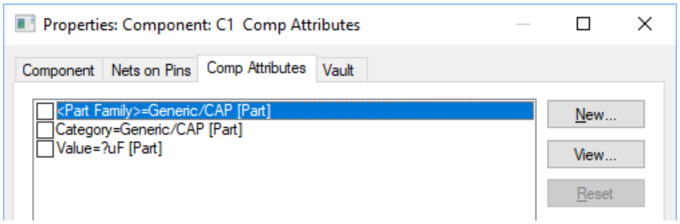
Optional parameters allow for the override of the options specified from within Pulsonix. By running Pulsonix from the command line with a Pulsonix Command File as a parameter, the Vault Library Export facility can be utilised from the Windows Scheduler to create a regular reoccurring task. The full recommended command line is shown below:

```
Pulsonix.exe -hidden -commandfile <path to command file>
```

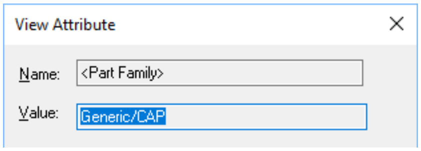
This feature was back-fitted to 10.5

Attribute Properties – View Name and Value for Protected status

A change to the **Properties** dialog and **Attributes** page now means you can select a protected Attribute and Value and use the **View** button to view it (instead of Editing it). Previously, it wasn't possible to view it because it was protected and not 'editable'. The **Edit** button changes state between **Edit** and **View** depending on the attribute status.

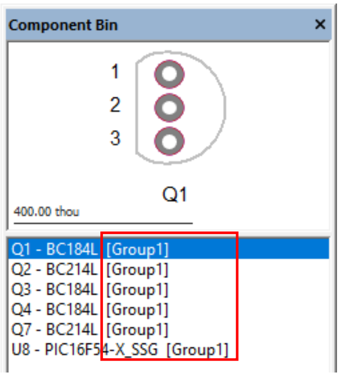


The feature enables you to copy and paste the Attribute name or Value using standard Windows commands from the dialog presented.



Group Names Displayed in Component Bin

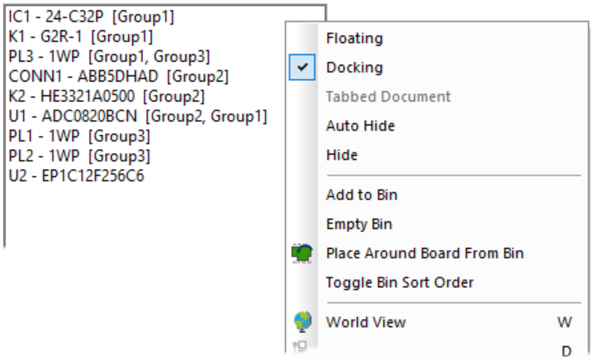
Group names are now displayed in the **Component Bin**.



Toggle Bin Sort Order in Component Bin

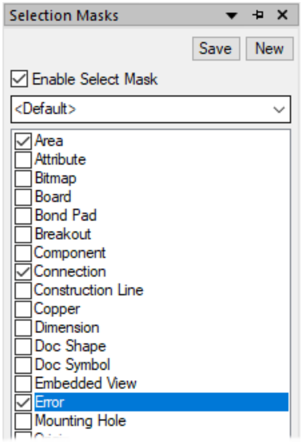
When using the **Component Bin**, a new option, **Toggle Bin Sort Order**, has been added to the context menu. The current sort order uses Component Name, this new toggle allows you to also sort by group name. By default, the sort order will use Component Name.

When sorting by group name, if an item is in multiple groups then its first group will be used. If multiple items are in the same group, they will be sorted by their Component Names. All items that are not in a group will be sorted by their Component Names and will appear after all the items in groups.



Double-Click selection in Selection Mask Bar

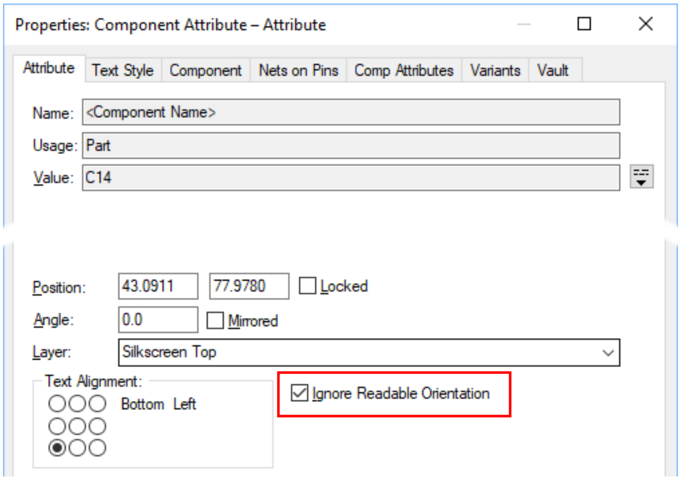
When in the **Selection Masks bar**, you can now double-click on a mask category check box which will deselect all currently selected categories and then only select the category on which you have chosen.



In the above example where three different categories are selected, if say, Board is double-clicked, only Board will be selected and all other categories deselected.

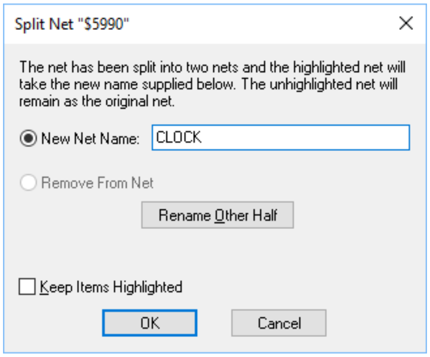
Override Readable Orientation of Text

The **Ignore Readable Orientation** check box has been added to **Text Properties** and **Attribute Properties** dialogs. Selecting this switch disables the effect of the **Adjust To Readable Orientation** switch (from **Design Settings** – **General** dialog) for that specific text.



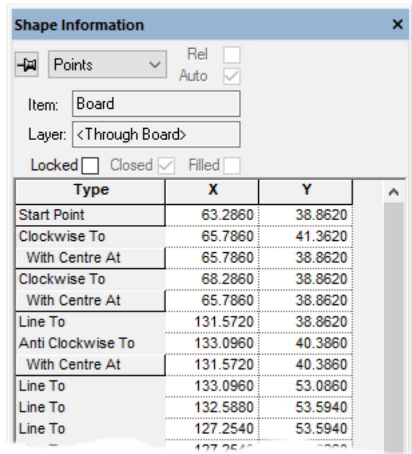
Split Net Highlights

There is a new check box on the **Split Net** dialog, **Keep Items Highlighted**. This means that when a net is split, if a resistor component is dropped on to the net for example, then no net highlight will take place. The net highlight is used to indicate the renamed portion of the net. This mode is off by default and can be selected to on for use in the option if and when required.



Shape Information/Editing Bar

A new Shape Information Bar has been added to Pulsonix. This appears as a dockable window accessible from the **View** menu and displays detailed information about the currently selected shape in a grid format that also updates on the fly. It is available to use when editing PCB and Schematic designs, symbols and technology files. The text in the grid describes the shape details that may be edited allowing precise modifications to be made.



Most check boxes relate to existing **Properties** and are self-explanatory.

The **Auto** button will cause the Shape Bar to automatically switch to the most appropriate mode to display a selected shape, i.e. if a circle is selected it will be displayed in **Circle** mode. Polygon shapes will be shown in the most recently used of the possible modes.

Shapes drop down list

The shape information can be displayed in various formats dependent on the current mode selected from the dropdown list. When a different mode is selected the grid columns will reconfigure to display the chosen format. In all modes, shape points are shown as pairs of coordinates in adjacent X and Y columns in the grid. The various modes are described below:

Points

The shape is shown as an initial Start Point and a series of Lines or Clockwise/Anti-clockwise arcs to given points. For arc segments, the arc centre point is shown on the next row. The points will be shown in absolute or relative coordinates depending on the **Rel** setting.

Point Offsets

The shape is shown as an initial Start Point and a series of Lines or Clockwise/Anti-clockwise arcs to given points expressed as offsets from the previous point. For arc segments the next row shows the centre point for the arc as an offset from the previous point. Only the Start Point will be shown in absolute or relative coordinates depending on the **Rel** setting.

Segments

The shape is shown as a series of segments starting from the given point with the given length and arc angle. The end point of the segment is given by the start point of the next segment. For the final segment of an open shape there is an additional row showing its end point. The direction of arcs is given by the sign of the angle with a negative value indicating an anti-clockwise arc. The points will be shown in absolute or relative coordinates depending on the **Rel** setting.

Segment Offsets

The shape is shown as a series of segments, similar to Segment mode, but with their start points expressed as offsets from the previous point. Only the start point of the initial segment will be shown in absolute or relative coordinates depending on the **Rel** setting.

Circle

If the selected shape is a circle it is shown as a Centre Point and a Radius. For any other type of shape, the display will be blank therefore this mode is best used with the **Auto** option enabled.

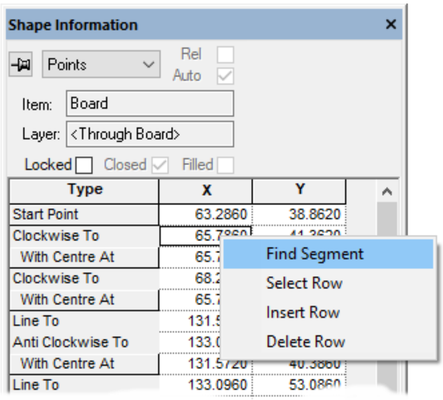
Rectangle

If the selected shape is a rectangle it is shown as a Start Point and a Size; **X** = width and **Y** = height. For any other type of shape, the display will be blank therefore this mode is best used with the **Auto** option enabled.

Note, only orthogonal rectangles are recognised by the Shape Information Bar.

Context Menu Options

When a shape is selected, you can **right click** on a row in the grid of the shape bar and click **Find Segment**.



This feature shares the **Find Bar** options (**Options, Find, Action On Found Item**) for **Highlight** and **Flashing** and will find the chosen segment in the design. If you have the **Highlight it** and **Flash it** boxes checked, then the chosen segment will be found, highlighted and/or flashed.

Find a segment using the grid

Right click the mouse over the row in the grid you with to find in the design. From the context menu select the **Find Segment** item. The segment will be located by altering the view to include it and then either selecting it, highlighting it, brighten it (dim all other items) or flashing it. The find action depends on the current **Find Options**. Find Segment is not available when using Circle or Rectangle mode.

Insert a new segment into a shape

Right click the mouse over the grid at the point you wish to insert a new segment. From the context menu select the **Insert Row** item. A new segment will be insert in to the shape at that point. The exact position of the inserted segment will depend on the current Shape Bar mode; in a Segments mode, the new segment is inserted from the centre of the existing segment to its end point. In a Points mode, the new segment is inserted after the end of the existing segment to the centre of the next segment (if there is one). In either case, the end point is gridded using a step size relative to the **Working Grid** . Insert Row is not available when using Circle or Rectangle mode.

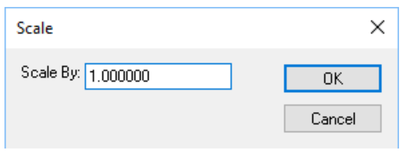
Delete a segment from a shape

Right click the mouse over the grid row containing the segment you wish to delete. From the context menu select the **Delete Row** item. The way the segment is removed depends on the current Shape Bar mode; in a Segments mode, the corner at the start of the segment is removed. In a Points mode, the corner at the end of the segment is removed. It is not possible to delete segments from a shape if doing so would cause it to become invalid. Delete Row is not available when using Circle or Rectangle mode.

Scale Option

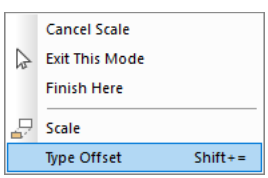
The **Scale** feature has been updated to allow for scaling on-the-fly of **Shapes**, **Text**, **Schematic Gates** and **Bitmaps**.

An item is selected in the design and the **Scale** option selected from the context menu.

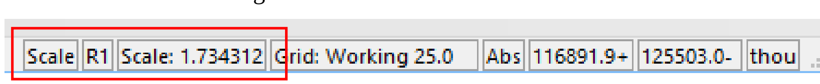


When scaling text, the ability to cycle through styles of the same width is available on the context menu using the **Next Style** option.

The Scale option can also be used as a 'mode'. Select the **Scale** option first from the **Utilities** menu, there a number of modes you can use:



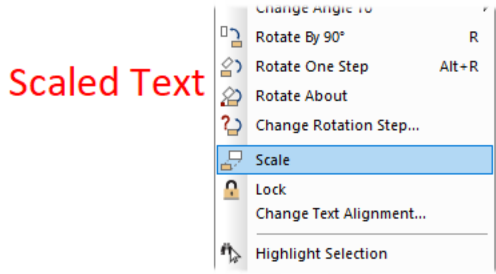
You can select an item and dynamically rescale it by moving the cursor. The actual scale is shown on the **Status** bar as the scale is changed.



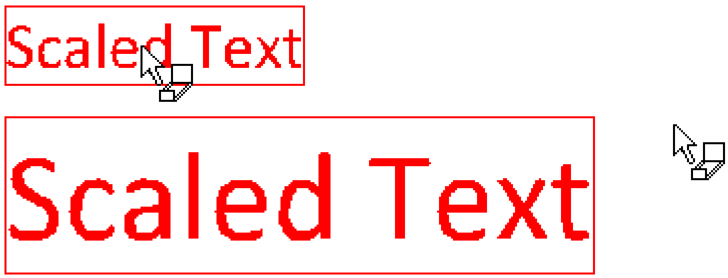
You can manually type in a **Scale** or **Offset** as a 'one shot' function to rescale an item.

Scaling Text, Layer Stack Previews, Drill Tables and Inserted Reports

Scaling text and non-symbol design items works slightly different in that when you scale these, it will run through each text style and next the next biggest one available. This is activated by selecting text and choosing Scale from the right-hand menu.

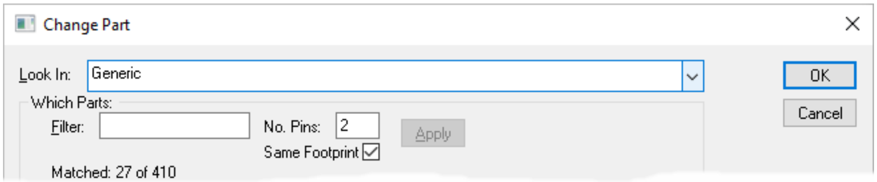


For example, each time the mouse is moved away from the text, the text is changed to the next size:



Change Part - Same Footprint check box

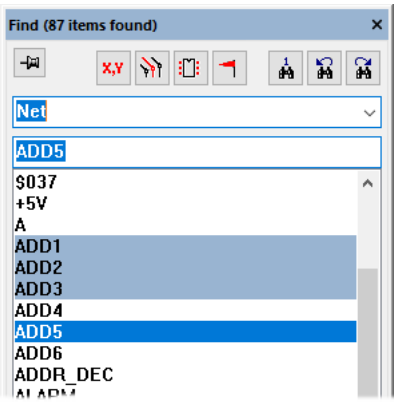
From the Change Part dialog in the filter section, you can now select the **Same Footprint** check box. This will allow you to filter the Parts to only contain ones with the same footprint as the current Part. If you are changing the Part of a Variant, then warnings will be displayed if the footprint or pin mapping of the newly chosen Part are different to the current part.



Find Bar Changes

Multi-Select in Find Bar

When you select several items in the **Find Bar** list using the **Ctrl key** they are all selected, but now they are **kept highlighted** in the list so you can see all the selections not just the current one.



Enhanced Selection

When using the **Find** bar, some list entries are used by more than one item in the design, for example **Group** and **Style** names. In this case, on selection in the Find bar, only the first item matching the search criteria will be found.

You can now hold the **Shift** key down when selecting these list entries to find all items matching the search criteria, not just the first one, for example, all items in the selected group.

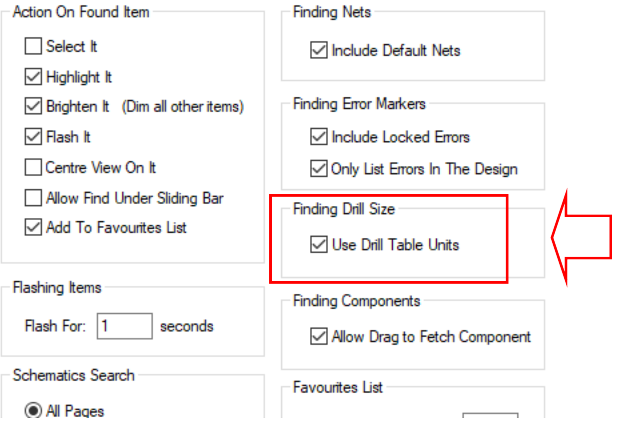
Using the **Ctrl** and **Shift** keys together you can find multiple whole items, for example, multiple **Groups**.

This feature was back-fitted to 10.5

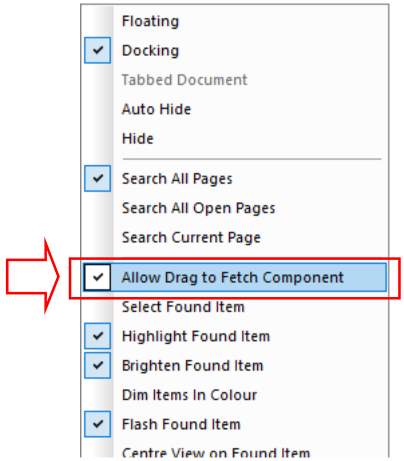
Fetch Component from Find Bar

Within the **Options** dialog and **Find** page, there is a new find option is available, **Finding Components – Allow Drag to Fetch Component**. This is set on by default. This allows you to drag a component from the find bar to “Fetch” it from elsewhere in the design, or from the bin.

In a Schematic design you will only be able to fetch a component that is on the current page.



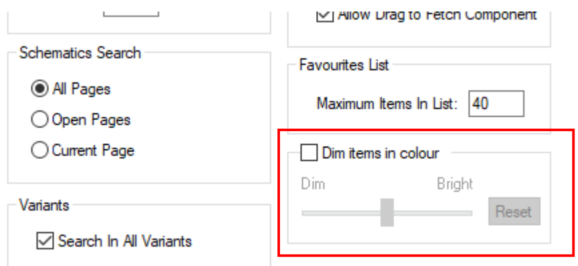
When displaying the **Component** category in the **Find Bar**, the **Allow Drag to Fetch Component** option is available on the context menu.



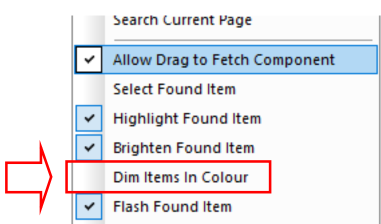
Dimming All Non-found Items in Colour

A check box has been added to the **Options** dialog and **Find** page to **Dim items in colour**. This will allow the Find bar to dim all the non-found items in colour, rather than all in grey.

A slider has also been added to allow the user to dim by an inputted amount.

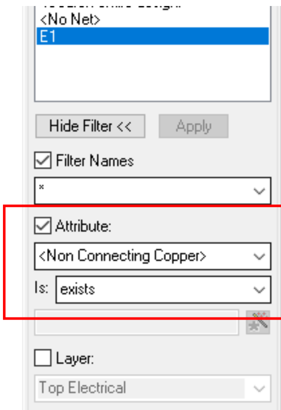


The **Dim items in colour** option is also available on the context menu on the Find bar, use it to toggle this mode.



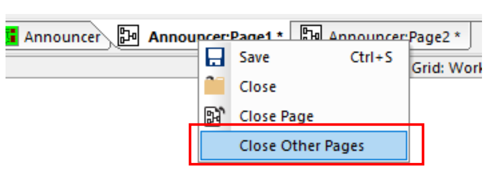
Search for Non-Connecting Copper

You can now search for non-connecting copper in the PCB editor using the <**Non Connecting Copper**> attribute filter in the **Find Bar**. By setting the search to **Is: Exists**, all Non-connecting Copper by net name will be listed.



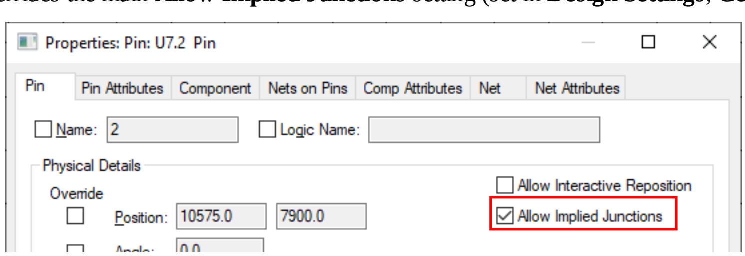
Close Other Pages (SCM)

A new command to **Close All Pages** has been added to the **File** menu and to the **Workbook** tab as a new command by right clicking on the tab. When applied, this works on all currently open Schematic pages except the current one.



Allow Implied Junctions Pin Override Switch (SCM)

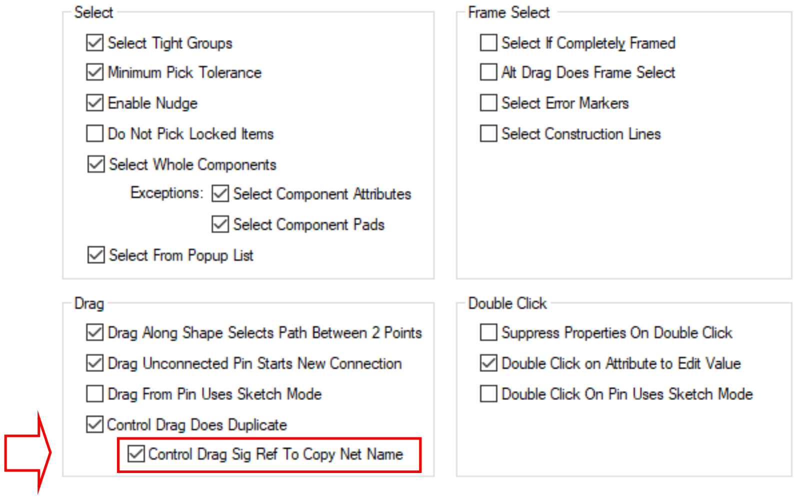
For a selected Pin in a Schematic design, on the **Pin Properties** dialog, there is a new switch for toggling **Allow Implied Junctions**. This enables you to set local Implied Junctions on individual pins and overrides the main **Allow Implied Junctions** setting (set in **Design Settings**, **General** page).



Copy Net Names on Sig Refs with Ctrl-Drag (SCM)

A change has been made when using **Ctrl-drag** (to duplicate) a **Signal Reference** where the symbol has a user-defined Net Name. Using Ctrl-drag now copies the **Net Name** also. If you use Ctrl-drag on a Sig Ref symbol that has a default system Net Name allocated, only the symbol will be copied.

By default, this mode will be on. If you wish to disable it, use the **Options** dialog and **Select** page and deselect **Control Drag Sig Ref to Copy Net Name** under **Drag** options.



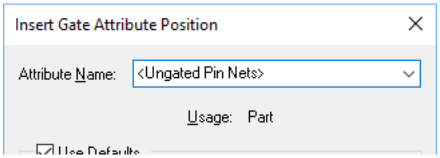
You can **override** this option locally by using **Ctrl-Shift-drag**. If the option is enabled then ctrl-shift-dragging will not copy the net name, and if it is not enabled control shift dragging will copy the net name.

This feature was back-fitted to 10.5

Insert Attribute Position (SCM)

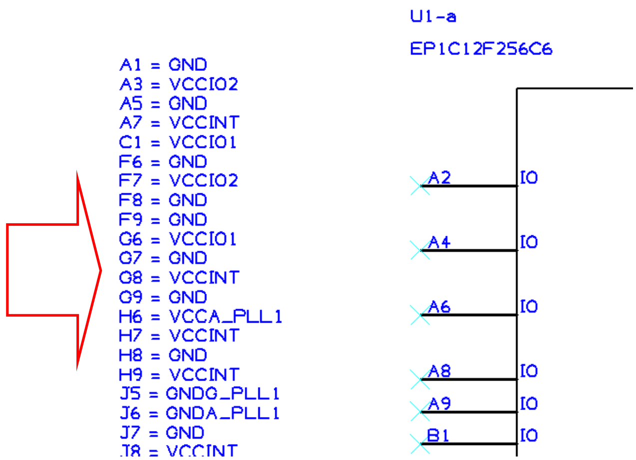
For a Component in a Schematic, there is a new **Attribute Position** attribute available called **<Ungated Pin Nets>**. This is available for a selected Component in the design using the **Insert Attribute Position** option.

When used, this will resolve to a list of Pin = Net for all nets assigned to ungated pins, thus allowing you to easily display the connectivity on the component that would otherwise be 'hidden' because the pins are not present on the symbol.



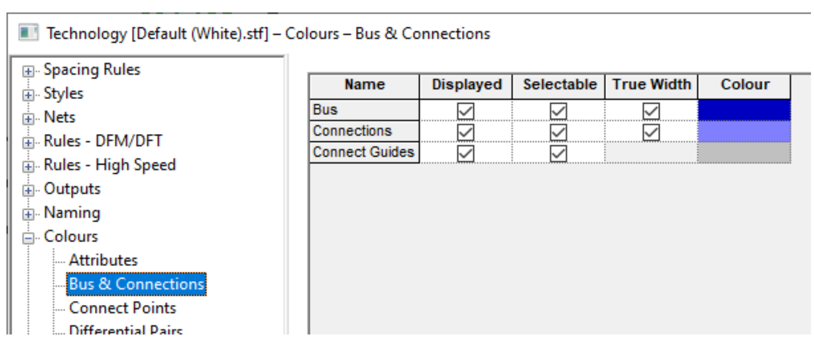
Previously, the only way to see ungated pin nets on the drawing was to insert a user report such as the provided **Ungated Pins Report** that makes a table of Parts, Components and their ungated pin nets that you can place on the drawing. With this new attribute value, you can display a simple list of those pins and nets right alongside the Component itself.

Inserted into the design it looks like this:



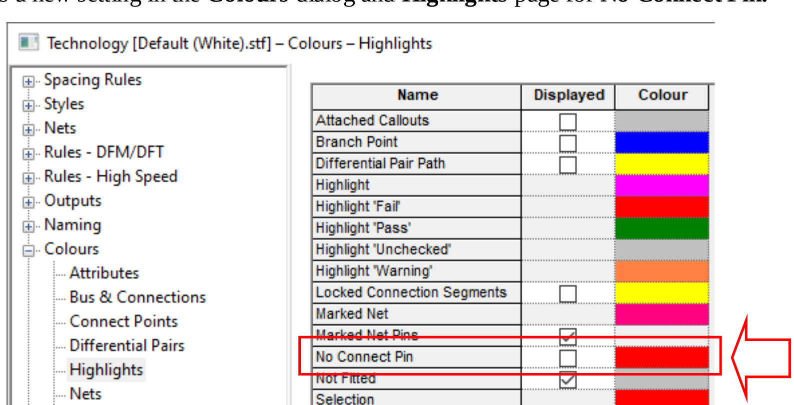
New Colour Category for Bus & Connections (SCM)

A new colour category for **Bus & Connections** has been added to **Schematic Colours** dialog. This brings together colours from **Others** and **Shapes**.



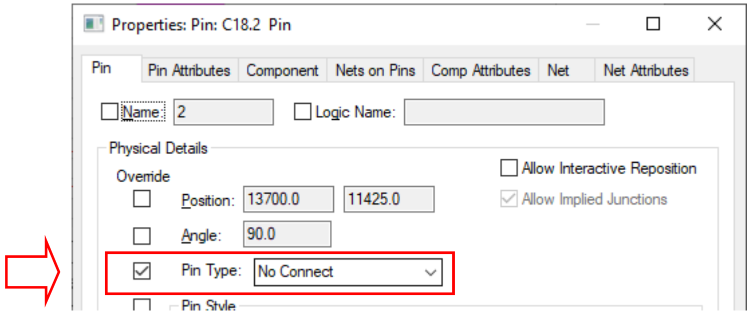
No Connect Pin Highlight Colour (SCM)

There is a new setting in the **Colours** dialog and **Highlights** page for **No Connect Pin.**

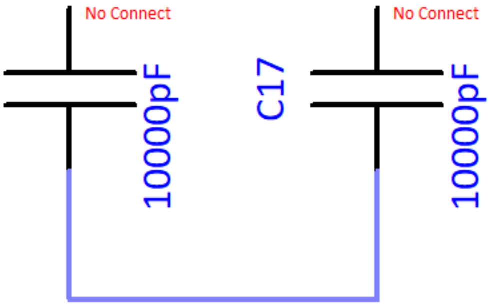


When enabled from the **Pin Properties** dialog, this causes the name of the pin type to be substituted into the Net Name attribute for No Connect Pins. Once enabled, a Net Name attribute position will be added to a pin if you make it No Connect and it will be drawn in the highlight colour.

This will also work on Component Pins and Block Ports.

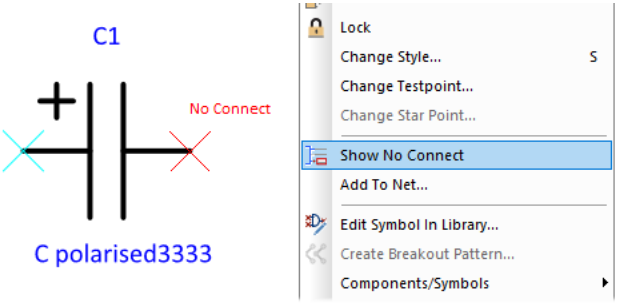


Displayed like this:



No Connect Pin context menu option to Show No Connect

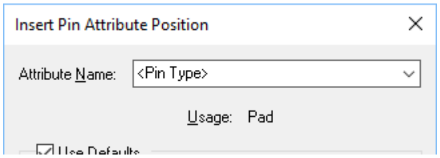
Within the **Colours** dialog, **Highlights** and **No Connect Pin** enabled, selection of a pin with a type of **No Connect** and right clicking, shows you the option **Show No Connect.** This replaces **Show Net Name** and will add a **Net Attribute Position** with the text **No Connect** value.



Display Pin Type Attribute (SCM)

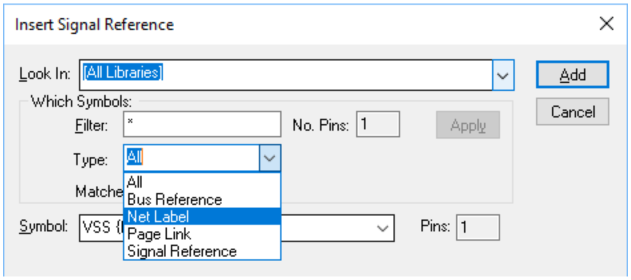
You can now display (in the design) the **Pin Type** on a pin using the **<Pin Type> attribute**. In either the symbol within the Symbol Editor or on the Component Pin in the design, use **Attribute Position** for a **selected pin** and choose <Pin Type>.

In the symbol, it will show as <Undefined> and in the design will be populated with the Pin Type if defined (locally in the design or in the Part definition).



Changes to Insert Signal Reference dialog (SCM)

The **Insert Signal Reference** dialog has been changed to now include all the Sig Ref symbol types available. This means the **Insert Bus Reference** and **Insert Page Link** options are no longer on the **Insert** menu. They can now both be added using the **Insert Signal Reference** option, (the new **Insert Net Label** feature will also be included in this dialog, see below).



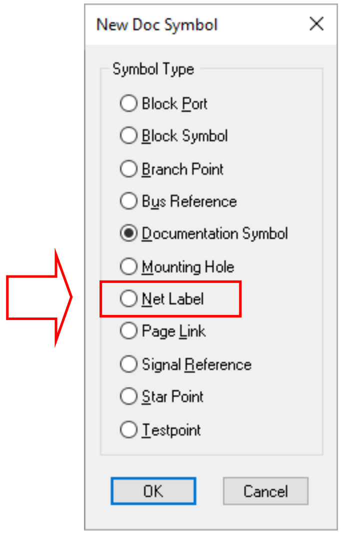
On use, you can either scroll through all the names and decide which symbol you require or you can use the **Type:** drop down list to refine the selection. For Type, you now have **All**, **Bus Reference**, **Net Label**, **Page Link** and **Signal Reference** available.

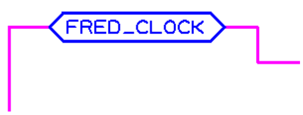
New Signal Reference Type - Net labels (SCM)

New Doc Symbol Type – Net Label

A new symbol type called **Net Labels** has been added. These are used to 'hang' on a net to explicitly show the net name on your Schematic design. Because it uses a special doc symbol, it means you can customise how it looks and where it is attached to. The symbol's net name is dynamic and will update as the name changes. On creation, the symbol will contain a pin and an attached callout with the net name attribute stored in the callout.

These can be created using the **New Doc Symbol** option from the **File** menu and **New** option by choosing the **Net Label** option or from the **Library Manager** and **SCM Doc Sym** page.

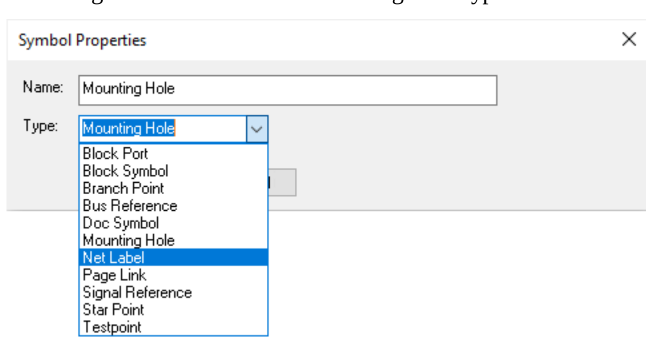




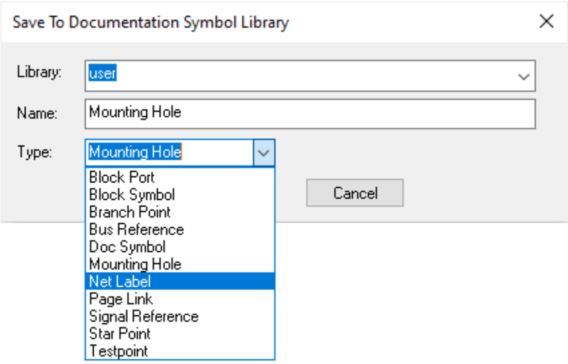
Once you've created the new symbol, use

Changing Existing Doc Symbols (Signal References) to Net Labels

To change an existing doc symbol to a **Net Label**, edit the Symbol in library and use the **Symbol Properties** dialog from the **Edit** menu and change the Type to Net Label.



This can also be done when using **Save To documentation Symbol Library**. The symbol will then need to be reloaded into the design.



If you don't do this, the **Next/Previous Symbol** option (see below) will still operate but will work on the current type. It may mean that it will spin through types that aren't relevant. Changing and reloading will make the Next/Previous symbol option more relevant and focused.

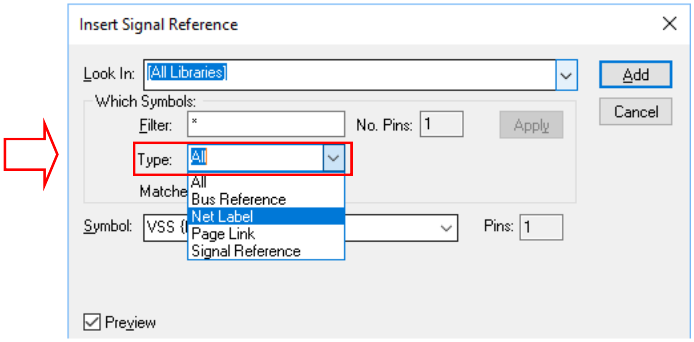
New Insert Net Label command

There is also a new **Insert Net Label** command available if required that can be added to your toolbar using the **Customise** dialog.

Inserting Net Labels into your design

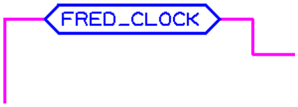
You can add net labels by using the **Insert Sig Ref** and selecting the **Type** as **All** or **Net Label**. Net Labels are used to 'hang' on a net to explicitly show the net name. Because it uses a special doc symbol, it means you can customise how it looks and where it is attached to. The symbol's net name is dynamic and will update as the name changes.

The **Type** drop down box on the **Insert Sig Ref** dialog enables the Sig Ref choice to be filtered. This will filter on **All**, **Bus Reference**, **Net label, Page Link**, and **Signal Reference** Doc Symbols.



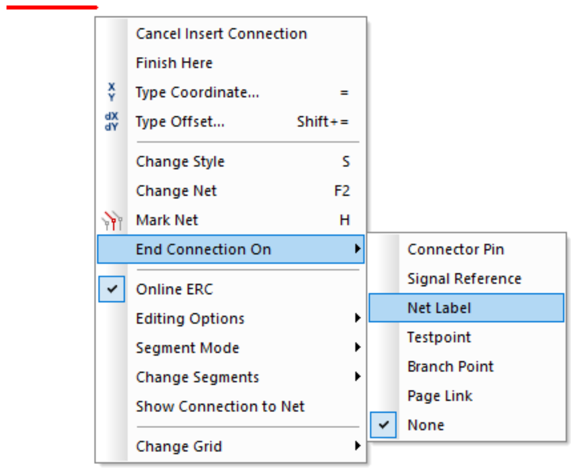
In all other **Insert Doc Symbol** dialogs, the **Type** drop down box will be read only and will display the type of the symbol you are inserting.

Once on the end of your cursor, drop the Net Label onto the net requiring it.



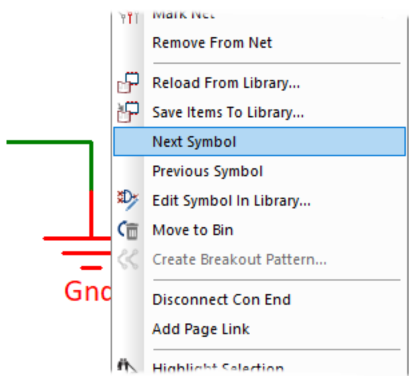
Insert Connection - Start/End on a Net Label

When inserting or editing a **Schematic Connection**, you can choose to **Start/End Connection On** a **Net Label** using the option from the context menu.



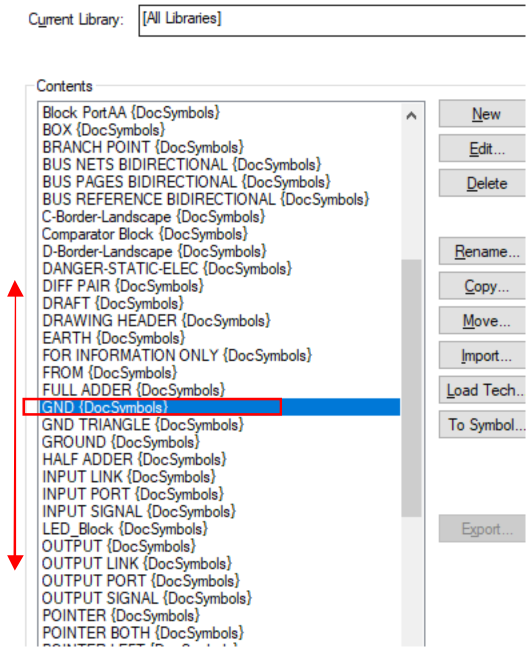
Next/Previous Doc Symbol Command

When a single **Signal Reference**, **Net Label**, **Bus Ref** or **Page Link symbol** is selected in the design, you can now select **Next Symbol** and **Previous Symbol** options from the context menu.



This will replace the currently used symbol with the next/previous symbol from your SCM Doc Sym Library. This will use Symbols of the same type.

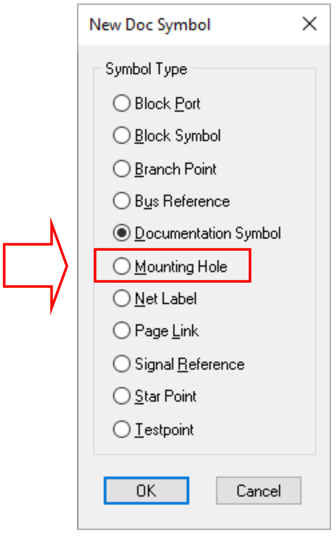
This means it will choose the next Net Label or next Signal Ref Symbol for example as relevant to the currently selected symbol type.

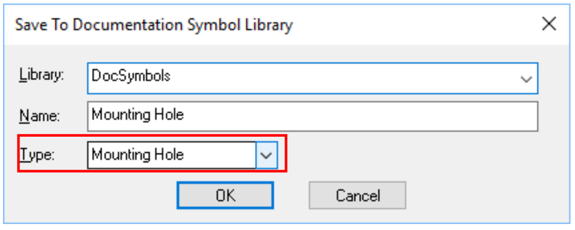


Mounting Hole Symbols in Schematic (SCM)

You can now add a **Doc Symbol** to a **Schematic** design to represent a **Mounting Hole** in the **PCB**.

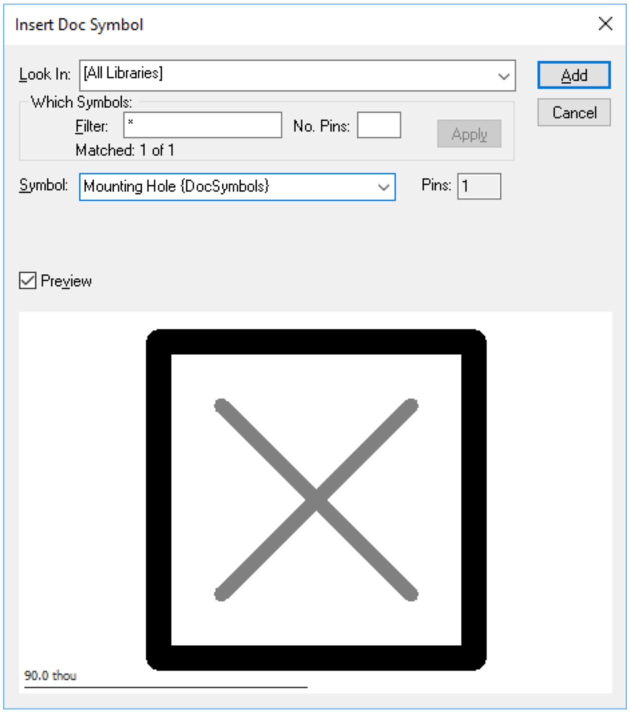
When creating a **New SCM Doc Symbol**, the choice now includes **Mounting Hole**. When saving the new Doc Symbol, you can choose **Mounting Hole** as the **Type**:





Adding The Mounting Hole Doc Symbol

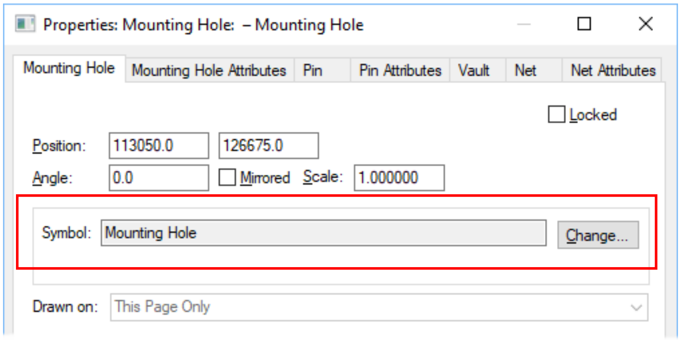
A new option on the **Insert** menu, **Mounting Hole**, enables this new type of doc symbol to be added to a Schematic design. Choose the Mounting Hole required from a list of items from your Doc Symbol library.



These symbols can be added to an existing net to enable their net connectivity in your PCB design.

Mounting Hole Doc Symbols In The Design

Once added to your Schematic design, the Mounting Hole Doc Symbol can be changed using **Properties**.



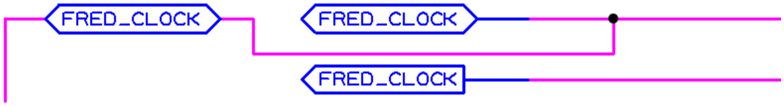
Following successful **Translation to PCB**, Mounting Holes are added to the **PCB** using the **Mounting Holes** setting in the **Design Settings**, **Defaults** page. This setting will select the layer for the Mounting Hole and Pad Style used, the same as if using the **Insert Mounting Hole** option.

Copy Net Names for Signal References (SCM)

Ctrl-drag, used to **duplicate a Signal Reference Symbol**, now also copies its **Net Name**.

This is particularly useful for a **Net Label Sig Ref** for example, where a Net Name should also be duplicated. It will only duplicate Net Labels where a user defined Net Name has been used, it will not duplicate auto-generated Net Names.

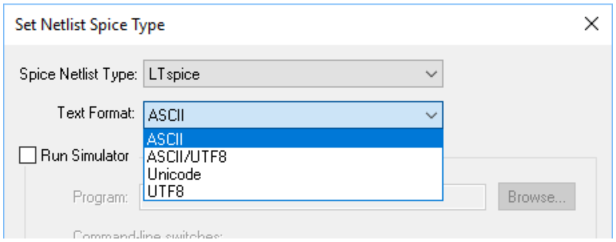
To disable this feature for use on another net, simply copy the Sig Ref symbol but press **Ctrl-Shift-drag** instead.



This feature was back-fitted to 10.5

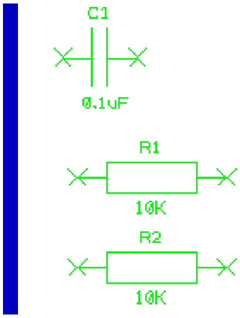
Text Formatting option for Spice Netlist Export (SCM)

When exporting a Spice netlist through the **Simulation** menu, on the **Set Spice Netlist Type** page, you can now choose how the Spice netlist handles international characters using the ASCII, Unicode and UTF8 settings. A setting can be chosen using the selections on the dropdown list. By default, this will be set to ASCII.

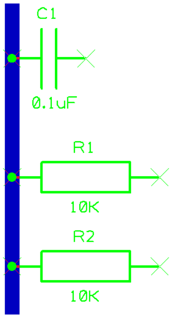


Weld multiple Components to a Bus segment (SCM)

The auto weld feature to drag the pins of a **Component** over a **Bus** segment has been extended to allow dragging of multiple Components over a Bus segment (like a set of resistors for example). This can be Components of a like type or dissimilar type.



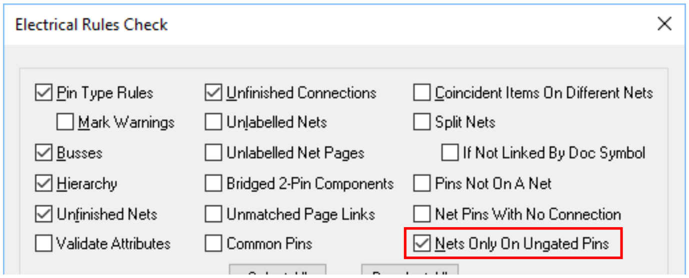
With weld markers shown >>



This feature was back-fitted to 10.5.

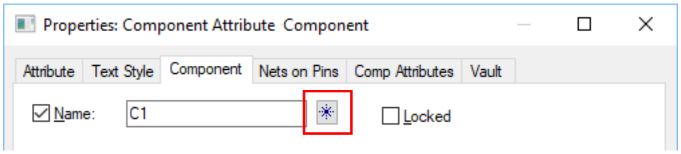
New ERC Check - Nets Only On Ungated Pins (SCM)

A new option in **Electrical Rules Check** to allow you to find which **nets** are only on **Ungated Pins**. If you have Components in your Schematic design that use Ungated Pins and these are not connected outside of this device, this check will identify that single pin nets have not been created when translating to the PCB.



First Free Component Name in Properties (SCM)

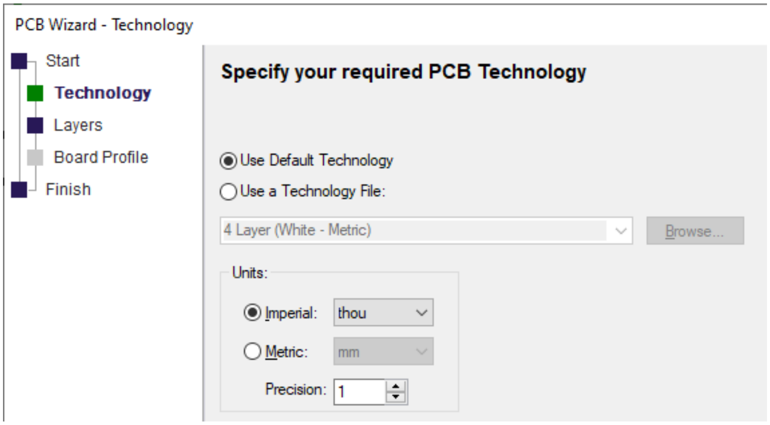
In Schematics, on the **Component Properties** dialog, there is now a button which will give you the first free component name that does not yet exist in the design.



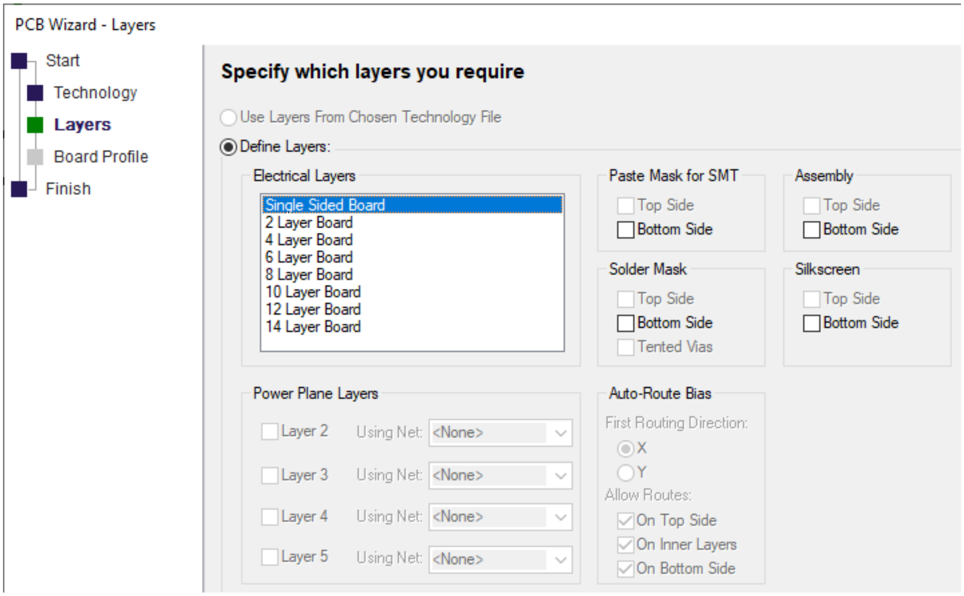
New PCB Wizard

A new wizard is available on the **New** dialog and **Wizards** for creating a new PCB design.

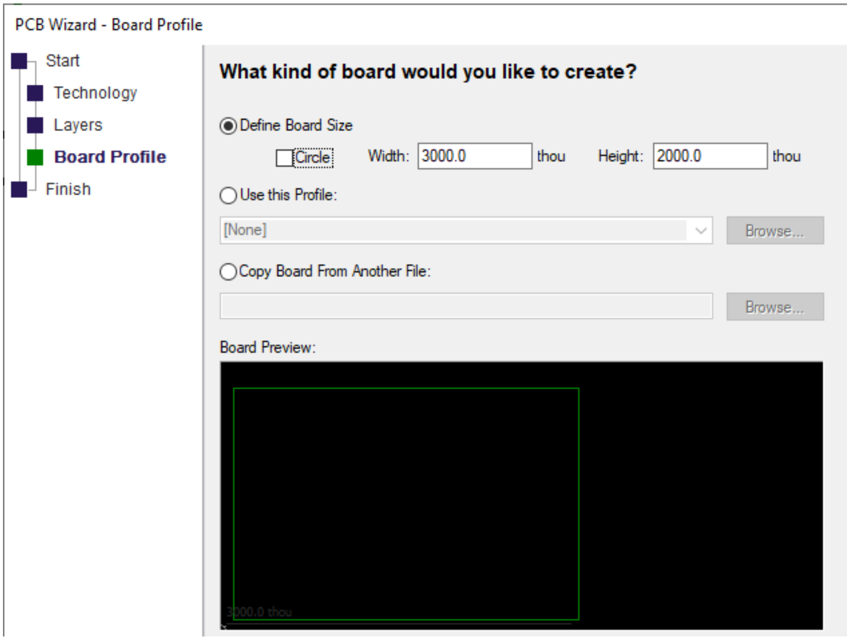
The **New PCB Wizard** allows you to specify the **Technology** file you want to use (or use the default technology) and the default units.



Then you can either use the **Layers** from the chosen Technology, or define your own using the type and bias required.

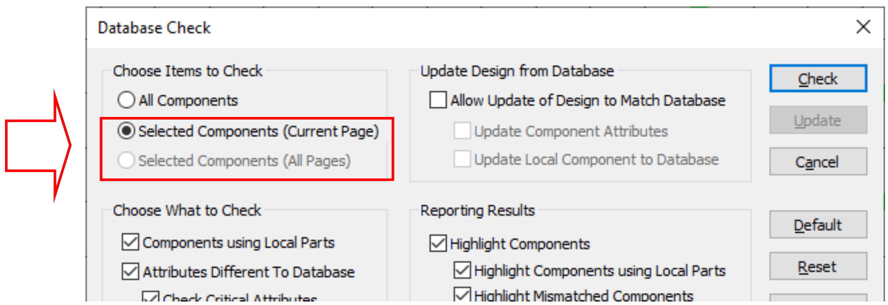


Finally, you can either define a **board** size, use a **Profile**, or copy a board from another file. Your selections will then create the appropriate PCB design once the design name is chosen on the **Finish** page.



Changes to Database Check and Update Options

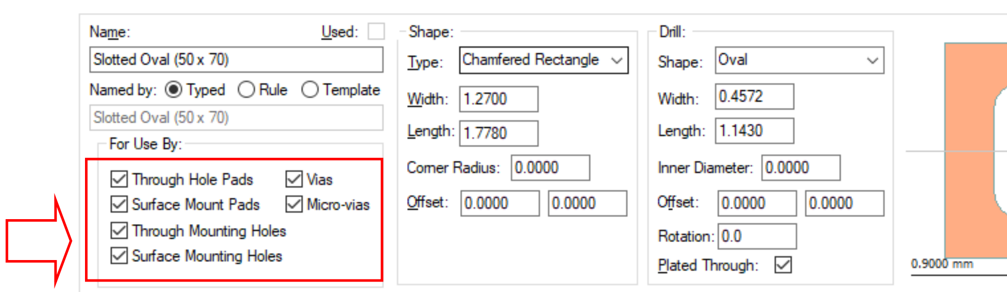
If you have the **Pulsonix Database Connection (PDC)** option, from the **Tools** menu and **Check/Update Against Database** option in the Schematic editor, you can now choose to update the selected Components on the **Current Page** or **All Pages,** (All Pages means components selected on different pages).



Technology Changes

Pad Styles – new Usage Types

Pad Style usage on the **Pad Styles** dialog in the **Technology** has been expanded to include use for **Through Mounting Holes** and **Surface Mounting Holes**.

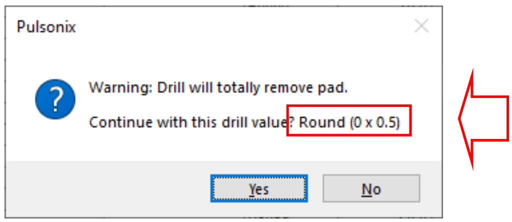


When adding Mounting Holes, if you swap from though-hole to surface mount and then change the Pad Style, if you have a long list of styles, this will aid distinguishing and refining the list to only use pads that are approved for this function.

This functionality was back-fitted to version 10.5.

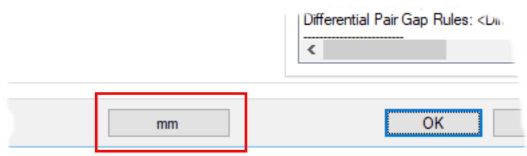
Drill Removes Pad Warning now shows Pad Style

When adding or editing **Pad Styles**, when you press the **OK** or **Apply** buttons on the dialog to exit it, if there are a number of pads that have a drill size larger than the pad size, the warning dialog presented will now also show you the Pad Style name so it can be easily identified. Previously, the Pad Style wasn't shown.



Units shown in Technology

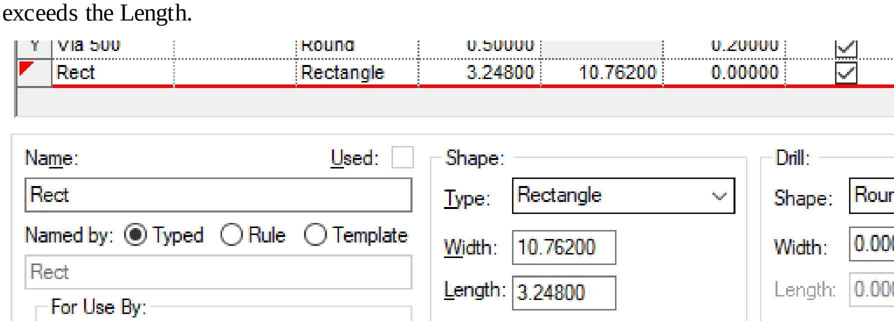
You can now temporarily switch the units on the **Technology** dialog using the Units button at the bottom of the page. The button will display the units currently being used on this page of the Technology. When toggled, a small asterisk * will appear to show you that the units currently being used are different to the design units. The setting of this button does not affect the design units.



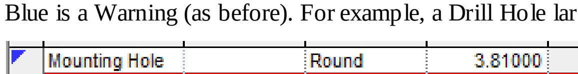
Additional Cell Status Indicators

The small triangle on the top left of the first cell in each row now indicates more states, Orange and Grey states have been added;

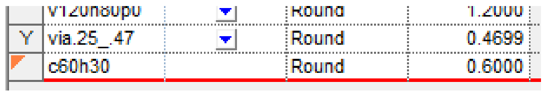
Red is an Error (as before). For example, on a Retangle Pad Shape where the Width defined exceeds the Length.



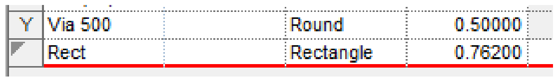
Blue is a Warning (as before). For example, a Drill Hole larger than the pad on a Pad Style.



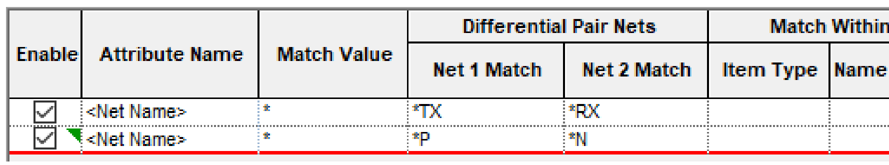
Orange indicates a new row



Grey is a modified row. This has been extended to all pages that have a **Used** or **Enabled** first column, e.g. **Styles**, **Layers** etc.



For **Rules** pages only, the Green triangle in the top right corner of the cell indicates the rule contains a note (as before).

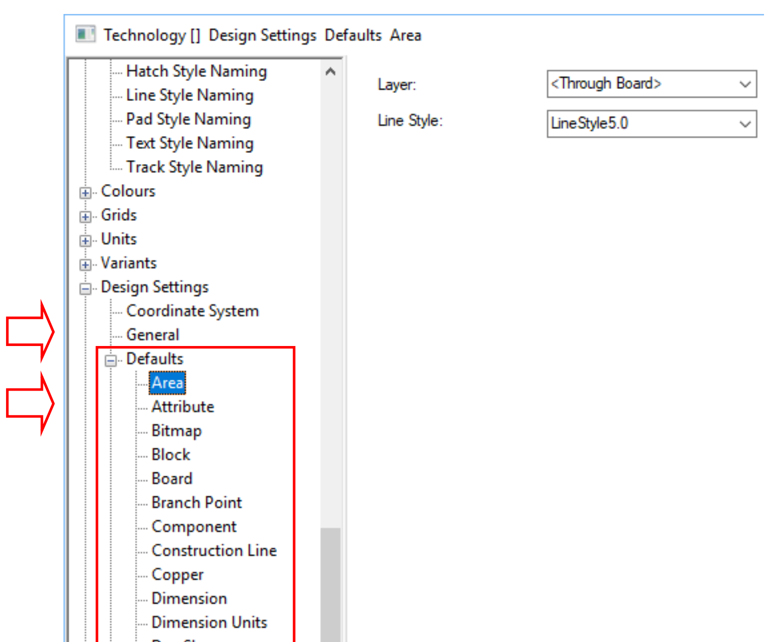


Updated Design Settings Pages

The **Design Settings Defaults** pages have been moved from a separate property sheet (accessed from General/Defaults in Technology) to a sub-category of **Defaults** off the **Design Settings** tab in the **Technology**.

The new *switch Units* button that has been added to the Technology dialog also works for all compatible **Defaults** pages, for example Branch Points.

The **Edit Default**" button that appeared on the **General/Defaults** page has been removed as it is no longer required and the page has been renamed to **General**.



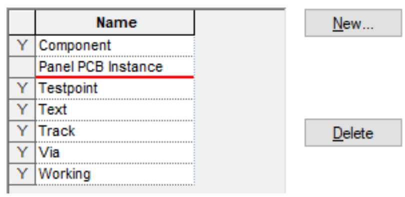
Changes to Grids dialog

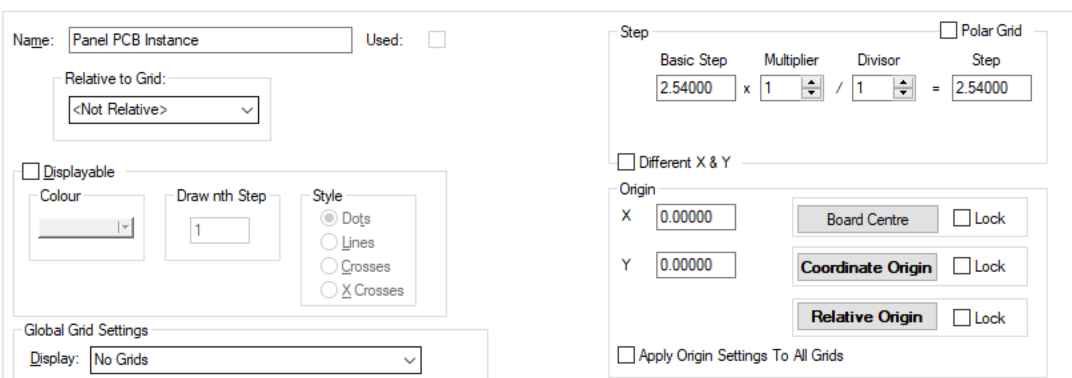
There are three main changes to Grids;

- Grid presentation in a table
- Removal of in-built grids
- New Tools Grid page

Grid Presentation

Design **Grids** are now presented in a 'grid' and operate in much the same way as other gridded pages in the Technology.



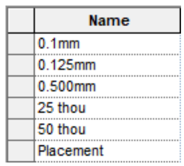


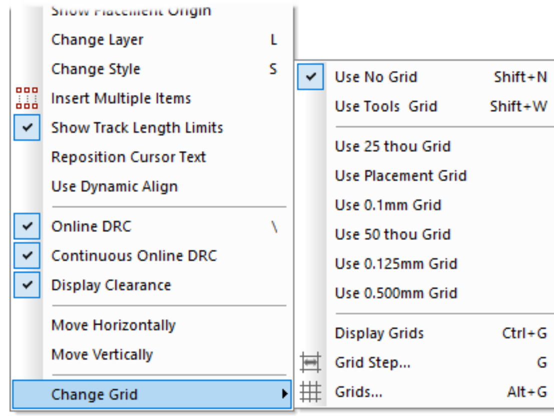
No In-built Grids

There are now no in-built grids, and you can rename and delete any unused grids.

Previously, built-in grids were shown in brackets, like <Working> for example. When your designs or Technology files are read into V11.0, the grid names will be preserved but you can now rename them or delete them (if unused). Like any table that has a value that is used shown with a **Y** next to its name, the name cannot be removed until unused. You can also remove all grid names from the list although this isn't advised. If all grids are removed, you will only be able to work gridless.

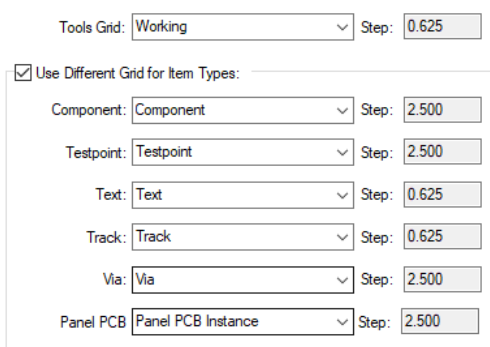
If you have allocated **shortcut keys** for existing commands **Next Grid** and **Next User Grid,** these will spin through all grids now. Any grids not required can be removed, for example, existing 'system' grids (in brackets such as <Working>).





New Tools Grid page

Under **Technology** and **Grids**, there is a new **Tools Grid** page. This is used to define the 'default' grids used by non-interactive items.



When the **Use Different Grid for Item Types** is selected, these grid settings are used:

Component: auto place, arrange

Test Point: auto place, auto insert TP, testability report, auto routers

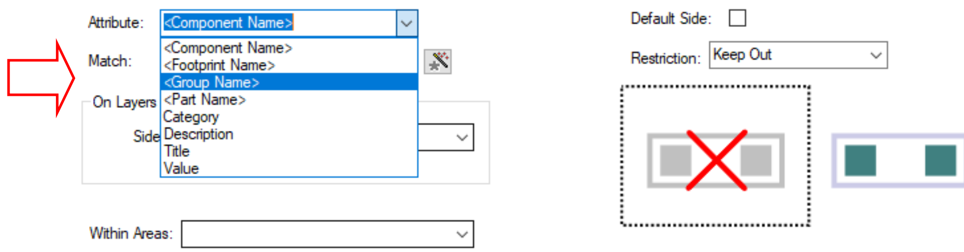
Track: auto routers

Via: apply vias, auto routers

PCB Panel and **Text**, these are currently not used.

Group Name available in Component Place Rules

You can now use **<Group Name>** as the rule attribute in the **Component Place** rules dialog in the Technology. This name is available on the **Attribute** drop down list.



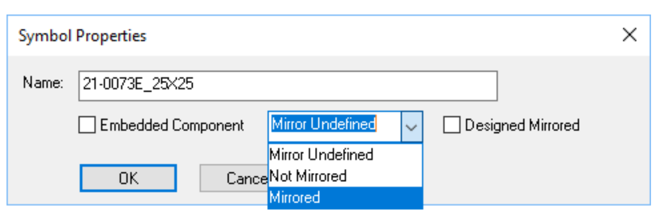
Component Placement Rule - Default Mirror State

There are new ways to define the default mirror state when adding a Component.

Footprint Editor – Mirror Status

In the **Footprint Editor**, using the **Edit** menu and **Symbol Properties**, you can define a footprint to be added **Mirrored** or **Not Mirrored**. This is used when adding a Component to a PCB design.

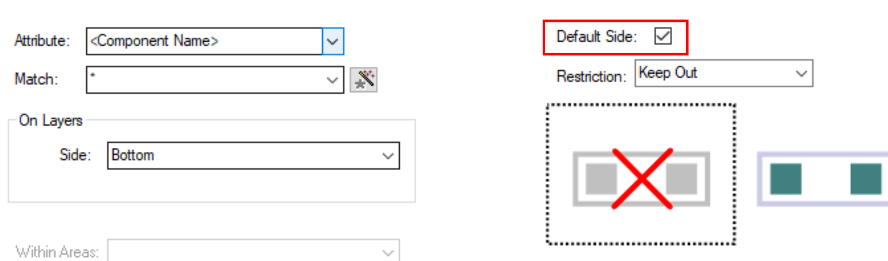
The normal state is **Mirror Undefined** - its mirror state is determined elsewhere. **Not Mirrored** means the footprint will not be mirrored, **Mirrored** means it will be mirrored.



You can define a footprint as **Defined Mirrored**, which means it is designed as it would appear when mirrored. See section below under *Define Mirrored Footprints*.

Technology Component Placement Rule

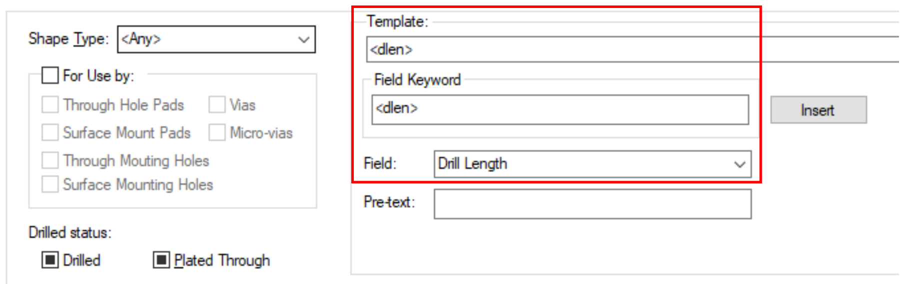
You can create a Component Placement Rule to default the side (i.e. the mirror state) based on Component Attributes.



By defining a rule based on a specific **Side** and selecting the **Default Side** check box, you can force the placement side when adding Components to your PCB design.

Pad Styles – New Naming Rule for Non-round Holes

You can now add drill length for non-round holes. A new **Field** of **Drill Length** is available for selection. A **Template** value of **<dlen>** is used when inserted.

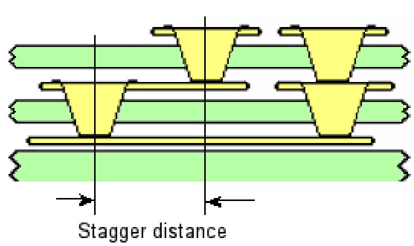


Micro-Via to Buried Via Stagger Spacing

Within the **Technology Spacing** rules and **Drill** tab of design level spacings, the **Stagger** option to has been renamed to **Stagger to Micro-via**.

A new spacing rule has been added to the **Drill** tab named **Stagger to Buried Via**. This allows for checking spacings between micro vias and buried vias on adjacent layer spans.

Each of these rules will be the minimum centre to centre distance between micro-vias on adjacent layer spans or the minimum centre to centre distance between a micro-via and a buried via on adjacent layer spans respectively.

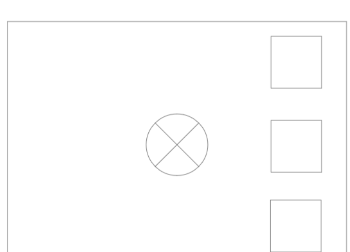


This feature was back-fitted to 10.5

User Defined Pad Shape Improvements

Define Pad Shape - Multiple Shapes Allowed

In **Define Pad Shape**, you can add multiple shapes for on a **Non-Electrical layer**. For example, to define multiple glue spots. This can be done by adding the extra shapes and changing their layer to an appropriate one.



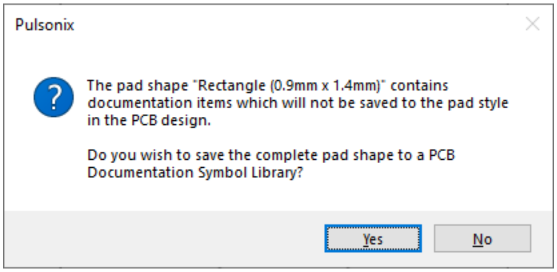
Define Pad Shape now contain Text, Dimensions and Callouts

When using the **Define Pad Shape editor** (when editing a pad style from a PCB Design) you can now add **Text**, **Dimensions** and **Callouts**. These are in addition to being able to add Construction Lines. **Properties**, **Technology Styles**, **Defaults** and **Colours** are available to support these documentation items.

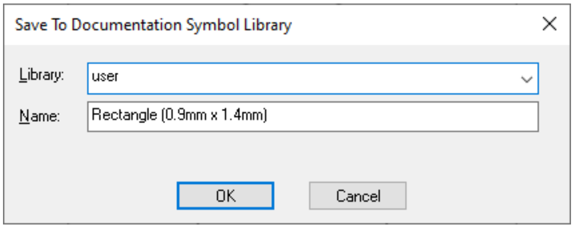
Saving Documentation items for reuse

In version 11, you can now save all of these items within the Pad Shape so that they can be recalled for use on future edits. Effectively, for special Pad Shapes that have this additional information you will create a set of Pad Shapes in your library. Previously, Construction Lines we transient and lost when you edited the Pad Shape editor. **To save these Documentation items, you must save the Pad to a PCB Doc Symbol in a library.** If you choose not to save the Pad to a Doc Symbol then the additional information will be lost. Not saving the Pad Shape will still mean it retains any user defined shapes as before.

When saving back to a style in the PCB design, if the Pad Shape is in a library or if the Pad Shape contains documentation items, you are asked if you want to save to a Doc Symbol library.



Choosing **Yes** will display the **Save To Documentation Symbol Library** dialog:



This can also be accessed from the File menu using the **Save To Library** option.

Choosing **No** will exit the Pad Shape editor, save the pad shape defined but will not save any of the additional documentation information, this will be lost.

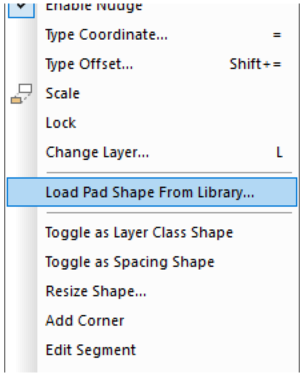
User defined pad shapes can now be added to the **Vault** as well as your **PCB Documentation Symbol Library**.

Saving a Pad Shape as a **Doc Symbol** will be saved as a new symbol type, **Pad Shape**. These can be edited the same as any other Doc Symbol and accessed from the **PCB Doc Symbol Library Manager**.

Load Pad Shape From Library

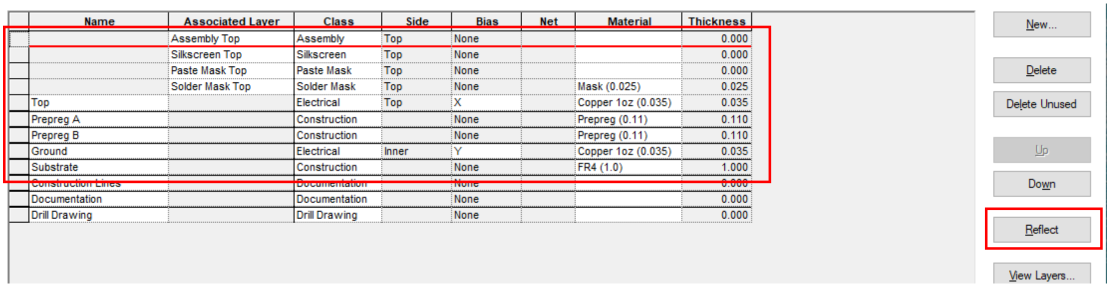
When editing PCB pad style, from in the **Footprint Editor** for example, if you wish to use a Pad Shape that has already been saved as a Doc Symbol, then for a selected pad, you can use **Load Pad**

Shape From Library from the context menu. If the pad style name is already in a library or vault you will be asked if you want to load it when first editing the style.



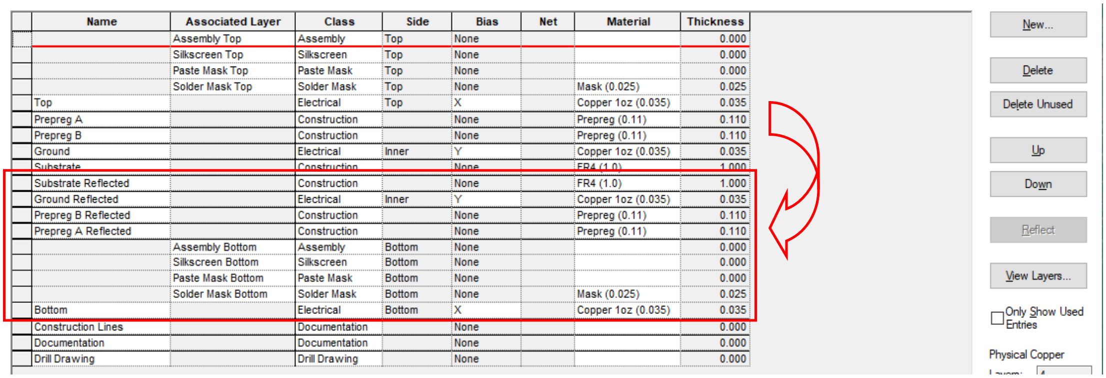
Technology Layers - Reflect Layers

A new **Reflect** button to the **Layers** page in the **Technology** has been added. This allows you to create the 'other' half of a layer structure by reflecting the current layers. For example, you can create the top 7 layers of a 14-layer board, and using Reflect, then create the bottom 7 layers automatically.



Name	Associated Layer	Class	Side	Bias	Net	Material	Thickness
	Assembly Top	Assembly	Top	None			0.000
	Silkscreen Top	Silkscreen	Top	None			0.000
	Paste Mask Top	Paste Mask	Top	None			0.000
	Solder Mask Top	Solder Mask	Top	None		Mask (0.025)	0.025
Top		Electrical	Top	X		Copper 1oz (0.035)	0.035
Prepreg A		Construction		None		Prepreg (0.11)	0.110
Prepreg B		Construction		None		Prepreg (0.11)	0.110
Ground		Electrical	Inner	Y		Copper 1oz (0.035)	0.035
Substrate		Construction		None		FR4 (1.0)	1.000
Construction Lines		Documentation		None			0.000
Documentation		Documentation		None			0.000
Drill Drawing		Drill Drawing		None			0.000

After using **Reflect**:



Name	Associated Layer	Class	Side	Bias	Net	Material	Thickness
	Assembly Top	Assembly	Top	None			0.000
	Silkscreen Top	Silkscreen	Top	None			0.000
	Paste Mask Top	Paste Mask	Top	None			0.000
	Solder Mask Top	Solder Mask	Top	None		Mask (0.025)	0.025
Top		Electrical	Top	X		Copper 1oz (0.035)	0.035
Prepreg A		Construction		None		Prepreg (0.11)	0.110
Prepreg B		Construction		None		Prepreg (0.11)	0.110
Ground		Electrical	Inner	Y		Copper 1oz (0.035)	0.035
Substrate		Construction		None		FR4 (1.0)	1.000
Substrate Reflected		Construction		None		FR4 (1.0)	1.000
Ground Reflected		Electrical	Inner	Y		Copper 1oz (0.035)	0.035
Prepreg B Reflected		Construction		None		Prepreg (0.11)	0.110
Prepreg A Reflected		Construction		None		Prepreg (0.11)	0.110
	Assembly Bottom	Assembly	Bottom	None			0.000
	Silkscreen Bottom	Silkscreen	Bottom	None			0.000
	Paste Mask Bottom	Paste Mask	Bottom	None			0.000
	Solder Mask Bottom	Solder Mask	Bottom	None		Mask (0.025)	0.025
Bottom		Electrical	Bottom	X		Copper 1oz (0.035)	0.035
Construction Lines		Documentation		None			0.000
Documentation		Documentation		None			0.000
Drill Drawing		Drill Drawing		None			0.000

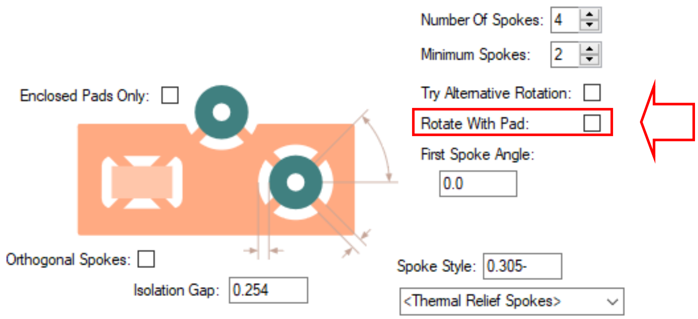
If, in your existing layer structure, you have a substrate already defined, this will also be reflected. Any additional unwanted and unused layers can be deleted as required.

The Reflect button will be disabled if you add or delete any layers from the grid and are yet to click apply (this could cause issues with the layer order). The button will enable once the changes are applied to the design.

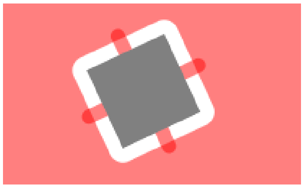
If you use Reflect and you have (for example) a Silkscreen Top layer and you already have a Silkscreen Bottom layer that is used in the design, no changes will be applied to that bottom layer. If the Silkscreen Bottom layer is not used in the design, then the layer data from Silkscreen Top will be copied to it (for example, Silkscreen Top is reflected and that layer replaces the current Silkscreen Bottom layer).

Thermal Rules – Rotate with Pad

Within the **Thermal Rules** page of the **Technology** dialog, there is an option to rotate the thermal pattern to the angle of the pad (**Rotate With Pad**).



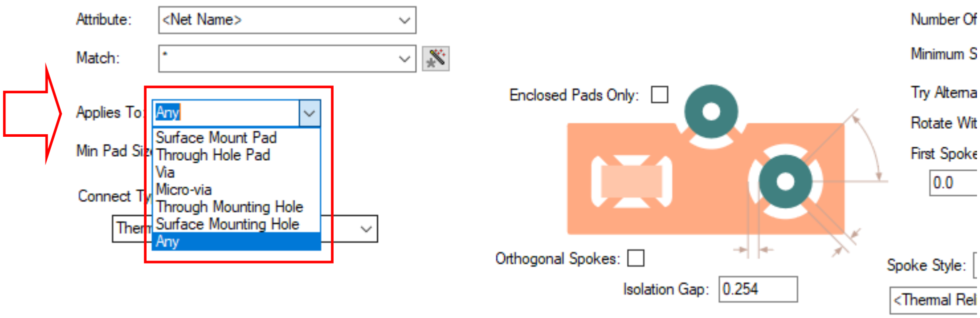
If a pad is at 25 degrees for example, and the first spoke angle is 0 degrees, the first spoke would actually be a 0 degrees relative to the pad, but 25 degrees relative to the board.



This feature was back-fitted to 10.5

Thermal Rules for additional Pad Types

You can now define explicit **Thermal rules** for **Micro-vias**, **Through Mounting Holes** and **Surface Mounting Holes**. To use this correctly, you should define the rules in the correct order required in order for them to take precedence. For example, Micro-via before Via (as via also applies to Micro-vias).

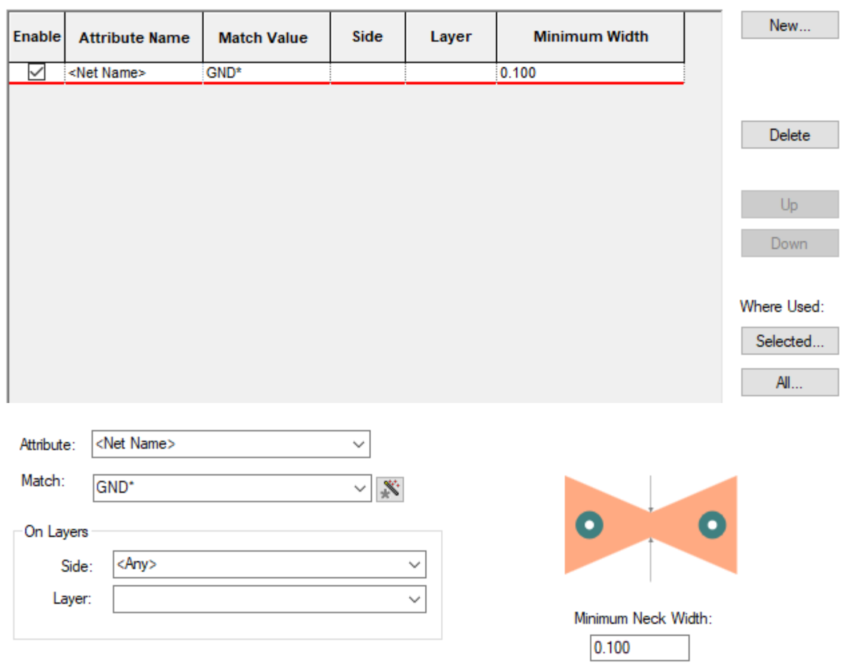


Copper Neck Width Rules (Power Dissipation)

The **Copper Neck Width Rules** dialog within the **Technology** is used to define the minimum necked width of a piece of copper which forms a connection on a net (this is not a slither check). The minimum necked width is the minimum width for the body of copper between two pins connected by that copper. The rule can be checked using a **Design Rule Check** and the rule under **Nets**, **Copper Neck Width.**

This rule could be used where **power dissipation** is important and you require a certain minimum gap between pads to be adhered to. It would be normal to check for specific nets to be adhering to this rule, such as power and ground.

This rule works on all poured copper but is **not** used for implied powerplanes (**Layers** defined in your **Technology** with a **bias** of **Powerplane**).

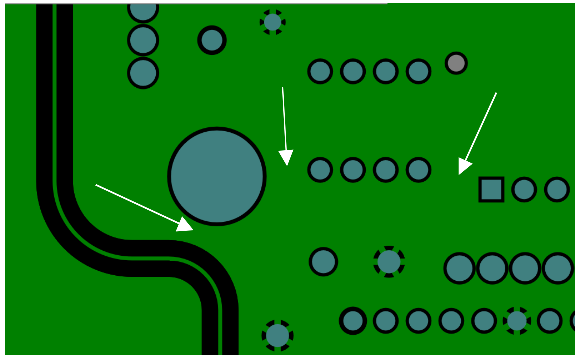


Minimum Neck Width

The **Minimum Neck Width** is the minimum width of copper between two pads connected by the copper. The width is the total of each piece of copper between cutouts in the path between the two pads. This will be positive closed copper shapes and doesn't include tracks or thermal spokes.

When the rule defines Pads, it means all pad types including, Component and free Pads, Mounting Holes, Vias and Doc Symbol pads provided they exist on the same layer as the check.

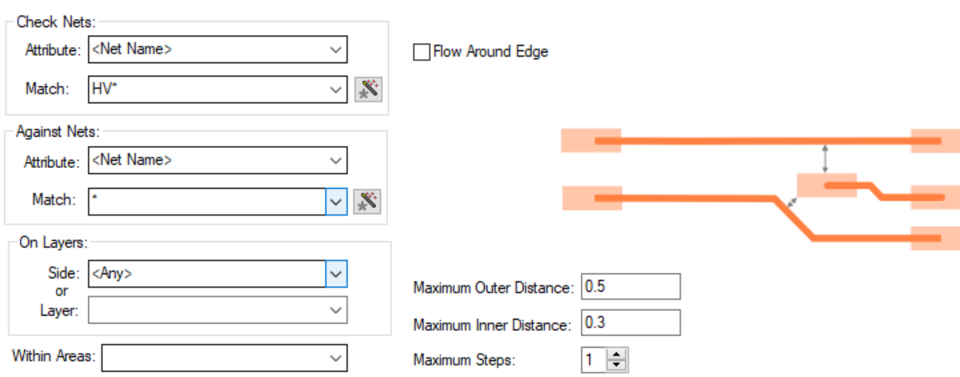
The illustration below highlights the ‘gaps’ that are available for power dissipation to take place.



Creepage Rules

A new page on the **Technology** dialog under **DFM/DFT**, named **Creepage Rules**, enables you to define rules for Creepage.

Enable	Check Nets		Against Nets		Check On			Creepage			
	Attribute Name	Match Value	Attribute Name	Match Value	Side	Layer	Area	Max Outer Distance	Max Inner Distance	Flow Around	Max Steps
☑	<Net Name>	GND	<Net Name>	*				0.5	0.3	☐	1
☑	<Net Name>	HV*	<Net Name>	*				0.5	0.3	☐	1



The purpose of the check is to report where the total surface gaps could allow creepage between two nets, either directly or through third party copper. You can also check creepage around board edges or cutouts. You can limit this check to a specific layer or named area.

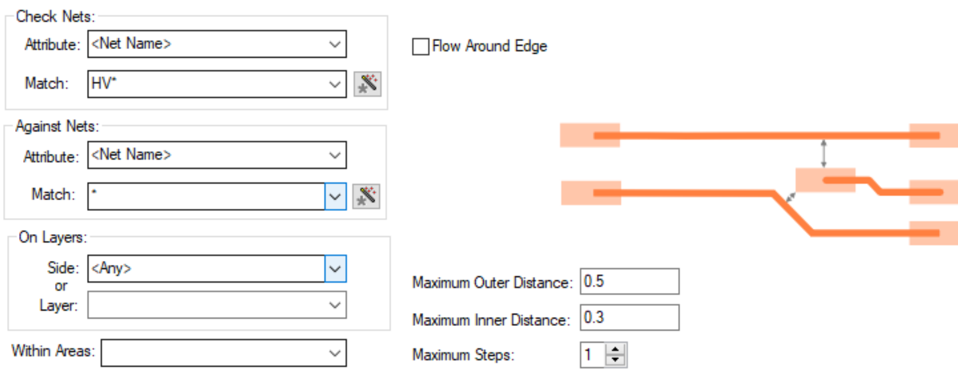
Creepage is caused by current flowing through pollutants on the surface of a board, this is typically increased with higher voltages. The creepage check finds gaps between critical nets between which current could flow. Third party copper can conduct current, meaning that the creepage can accumulate in several steps or jumps between the two critical nets. As this flow is across the surface of the board, the distance can effectively be increased by cutting a slot in the board, the creepage distance then follows the lip of the board edge. Current can also flow around the edge of the board in the third dimension, so changing layers. Although typically less of a problem, creepage can occur across inner board layers. There are therefore two values for the maximum creepage distance - Outer and Inner, which are then combined to give the final result.

Using this Rule

It is normal to define a specific net or a few nets that are critical to your design when considering creepage. These nets would be defined to be minimum distances from other nets in the design, possibly all others.

To achieve this, as with all other rules, you define the critical net (**Check Net**) and define the net which it is checked against (**Against Nets**).

Nets can be checked on a **Side**, **Layers**, **Layer sets** or within an **Area**.

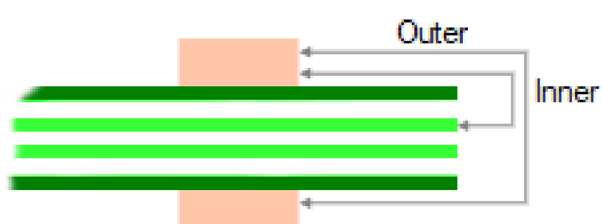


Maximum Outer Distance

The maximum total distance for which creepage would cause a problem on the outer surfaces of the board. This includes the distance around a board edge. This distance around the board edge can include changing layers, as well as around the lip of the edge. A value of 0 means that no creepage distance is checked on an outer layer.

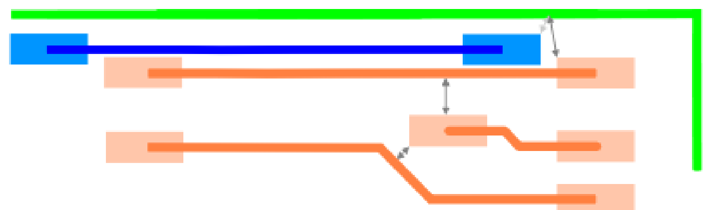
Maximum Inner Distance

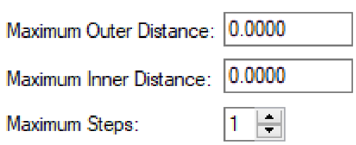
The maximum total distance creepage would cause a problem on inner layers of the board. This value is typically less than the Maximum Outer Distance. This does not include any layer changes as these can only be done on the outer edges of the board. A value of 0 means that no creepage distance is checked on an inner layer.



Flow Around Edge

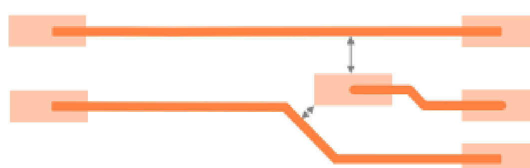
Check this box to allow layer changes by flowing around the edge of the board in the third dimension (the board edge includes board cutouts). This does not prevent flow around the lip of the board edge in the layer plane.





Max Steps

The maximum number of steps between copper. One step means a single gap between the two nominated nets. Two steps means that an intermediate piece of copper is traversed.

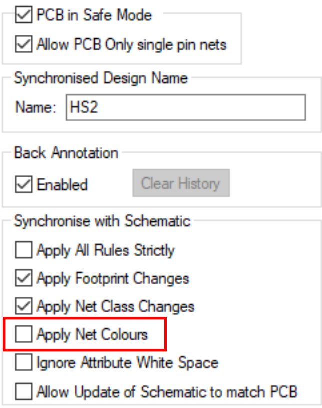


Combining Inner and Outer Distances

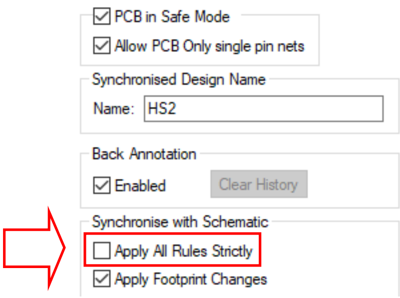
Each step will contribute to the total inner or outer distance. The inner and outer distances are combined in proportion to their respective maximums. If the total percentage does not exceed 100%, then there is an error. For example: Maximum Outer Distance = 2mm, Maximum Inner Distance = 1mm; actual total outer distance = 1.5mm (75% of maximum), actual total inner distance = 0.3mm (30% of total); total percentage = 105% so there is no error.

Design Settings – new Synchronise Design options

In the **PCB Technology** file under **Design Settings** and **Synchronise**, there is a new option switch in the list named **Apply Net Colours**. This will cause the **Own Colour** assignments for **Net Names**, **Differential Pairs**, **Signal Paths** and **Sub Nets** in the PCB to be synchronised with those defined in the Schematic. This may be useful for quickly identifying these items in the two designs.



Also, in this dialog, the Differential Pairs colour changes has been removed from strict synchronise option when the **Apply All Rules Strictly** switch is use.



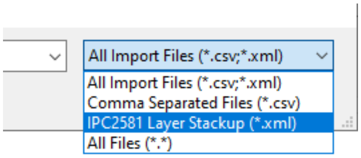
Import IPC-2581 Layer Stackup into Layers dialog

An option to import **Layers**, **Layer Classes**, **Materials** and **Impedance rules** from an **IPC-2581 Layer Stackup** file produced by 3rd party products, such as Polar Instruments has been added.

This format is now available from the **Technology** page and **Layers**, and also from the **Import Design Data** option on the **File** menu.

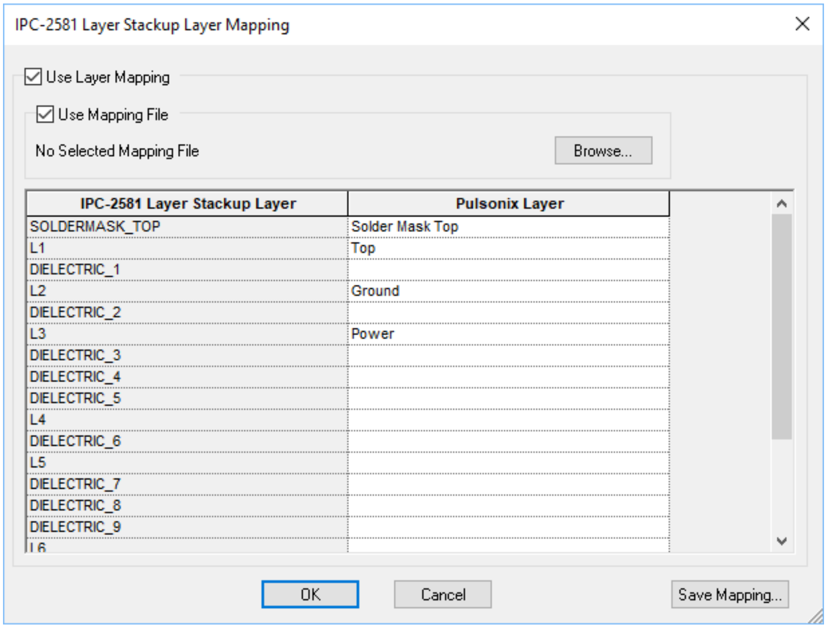
Layers Technology Page

On the **Layers Technology** page the **Import CSV…** button has been replaced with **Import…** When selected, you can import **CSV files** (as before) and new **Layer Stackup (*.xml)** files, (the .xml format is used by IPC-2581). The **Files of Type** filter allows you to choose the type required.



Import IPC-2581 Layer Stackup dialog

On selection of a valid IPC-2581 file you are presented with a dialog from where you can choose an existing mapping file or create a layer mapping set.



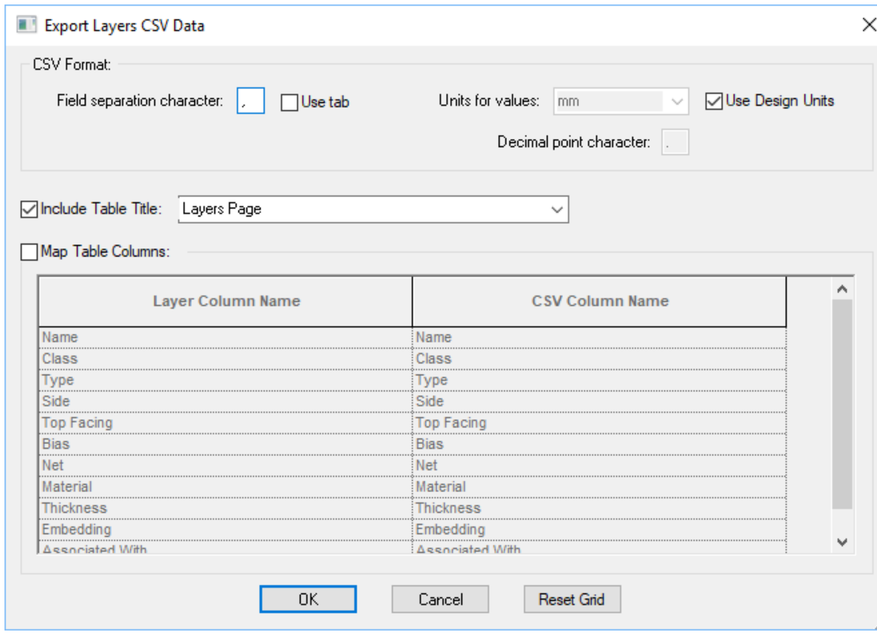
The imported file populates the **Layers** dialog:

Name	Associated Layer	Class	Side	Bias	Net	Material	Thickness
	Assembly Top	Assembly	Top	None			0.000
	Silkscreen Top	Silkscreen	Top	None			0.000
	Paste Mask Top	Paste Mask	Top	None			0.000
	Solder Mask Top	Solder Mask	Top	None		Mask (0.025)	0.025
Top		Electrical	Top	None		Copper Foil	0.036
Prepreg A		Construction		None		Prepreg (0.11)	0.110
Prepreg B		Construction		None		Prepreg (0.11)	0.110
DIELECTRIC_1		Prepreg	Inner	None		PrePreg 3113	0.060+
Ground		Electrical	Inner	None		FR4 Core Cu	0.053
Substrate		Construction		None		FR4 (1.0)	1.000
DIELECTRIC_2		Core	Inner	None		FR4 Core	0.075
Power		Electrical	Inner	X		Copper 1oz (0.035)	0.035
Prepreg D		Construction		None		Prepreg (0.11)	0.110
Prepreg C		Construction		None		Prepreg (0.11)	0.110
DIELECTRIC_3		Prepreg	Inner	None		PrePreg 1080	0.069
DIELECTRIC_4		Prepreg	Inner	None		PrePreg 7628	0.184
DIELECTRIC_5		Prepreg	Inner	None		PrePreg 1080	0.069
L4		Electrical	Inner	None		FR4 Core Cu	0.035
DIELECTRIC_6		Core	Inner	None		FR4 Core	0.300
L5		Electrical	Inner	None		FR4 Core Cu	0.035
DIELECTRIC_7		Prepreg	Inner	None		PrePreg 1080	0.069
DIELECTRIC_8		Prepreg	Inner	None		PrePreg 7628	0.184
DIELECTRIC_9		Prepreg	Inner	None		PrePreg 1080	0.069
L6		Plane	Inner	None		FR4 Core Cu	0.035
DIELECTRIC_10		Core	Inner	None		FR4 Core	0.075
L7		Electrical	Inner	None		FR4 Core Cu	0.053
DIELECTRIC_11		Prepreg	Inner	None		PrePreg 3113	0.060+
Bottom		Electrical	Bottom	None		Copper Foil	0.036
	Solder Mask Bottom	Solder Mask	Bottom	None		Mask (0.025)	0.025
	Paste Mask Bottom	Paste Mask	Bottom	None			0.000

Layers Import/Export CSV into Layers dialog

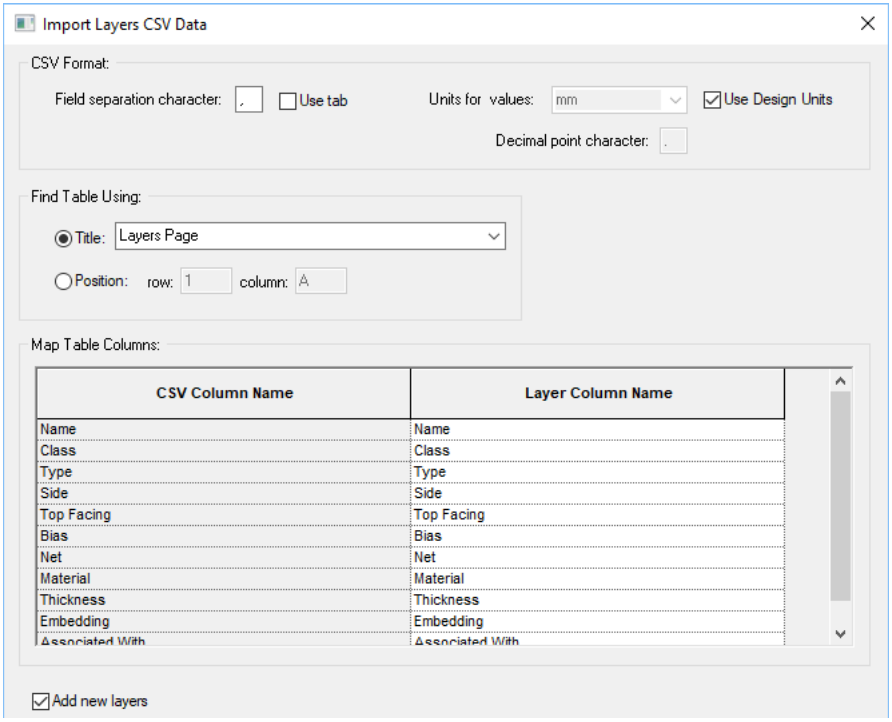
There are new options on the **Layer Technology** dialog that enable you to **Export Layers** to a CSV file and **Import Layers** from a CSV file. This is intended to allow you to create layer stacks quickly or from an external resource.

The export column headings for a layer are **Name**, **Class**, **Type**, **Side**, **Top Facing**, **Bias**, **Net**, **Material**, **Thickness**, **Embedding** and **Associated With**.



Import dialog

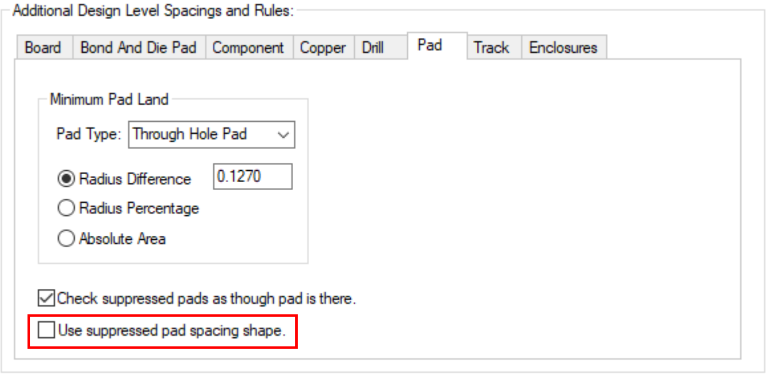
Similar to the Export dialog with controls for defining how the CSV file is formatted, any concessions for Row 1 and Column A and layer mappings available.



Option to use a spacing shape even when pad is suppressed

There is a new option in the **Technology** dialog, **Spacing Rules** and **Design Level – Pad** tab named **Use Suppressed pad spacing shape**.

With this enabled, it means that even when pad is suppressed (on an inner layer for example) as defined in your **Technology**, **Layers**, it will still use a spacing shape on a pad if one is defined. All other pad suppression will still be adhered to though if defined.



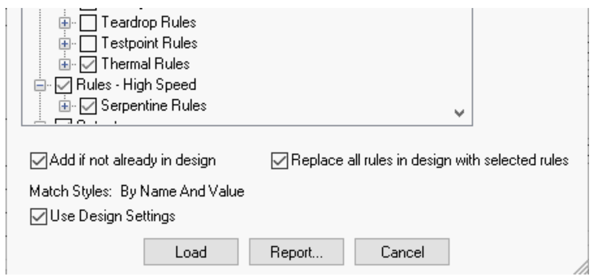
Load Technology - Matching Styles on Reload

Reporting on Reload

When loading a Technology file, a report will now be generated showing any styles in the design that have been changed as a result of the load.

Matching Styles on Reload

You can now view the current **style matching** method on the **Load Technology** dialog. This will be set to **By Name Only** by default but can be changed to **Use Design Settings'** using the check box. This will use the style matching method from the **Design Settings** dialog, **General** page and **Matching Styles** option.



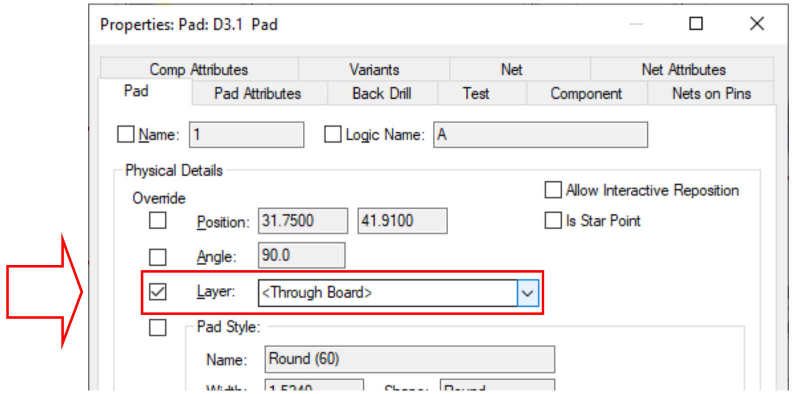
Rotated Pad Styles for 'Long' Pads

Long pad styles in Pulsonix are defined pointing up so the length lies along the Y axis. Some long pad styles in your existing designs are invisibly rotated by 90 degrees, so the length lies along the X axis. This situation normally happens when the design or library is imported from another system, or when using the **User-defined Pad Shape Editor** to define a rectangle or oval pad with that orientation. This has caused confusion as the rotation cannot be seen and there is no easy way to remove it.

A change has been made to show "Rotated" next to the pad width and length if it is rotated. Also, if the style is changed to a point shape (like round, square or octagon) the rotation is removed. This way a customer can change a rotated rectangle to a round and back to a rectangle to remove its hidden rotation.

Pad Properties Layer Override

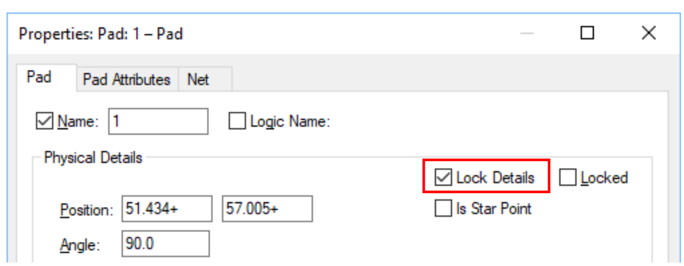
Added a layer override check box has been added to **Pad Properties** when Component Pads are selected in the design. This is useful if, for example, you locally swap a through hole pad Component Pad for a surface mount pad but then also need this to be on a specific <Side> layer, such a Top Side.



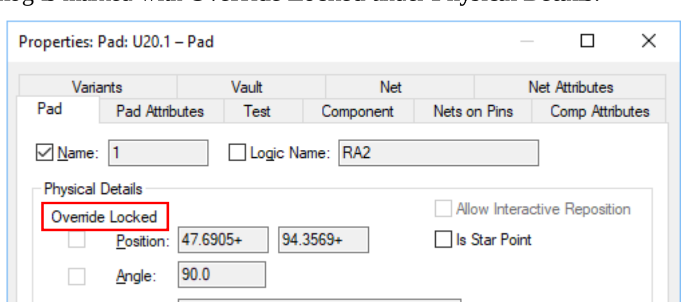
From an application perspective, a specific Part purchased is only available as through-hole and the board is mounted on a metal housing. If the connections of the high-voltage components were assembled as through-hole, this would produce a high-voltage flashover on the housing. Assembling the component as surface mounted on the top side, the board serves as insulation.

Lock Pad Details in Footprint Editor

Within the **Footprint** editor, for **Pad Properties,** there is now a **Lock Details** check box. This stops the physical details on a pad from being edited in the design.

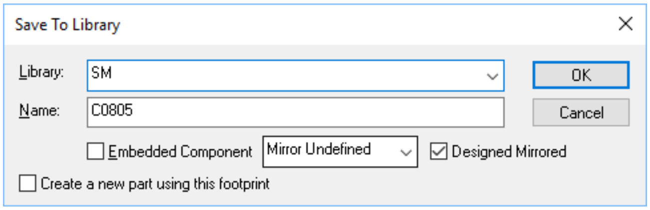


With this box checked, the pad on the footprint in the design is locked and its properties greyed out. The dialog is marked with **Override Locked** under **Physical Details**:



Define Mirrored Footprints in Footprint Editor

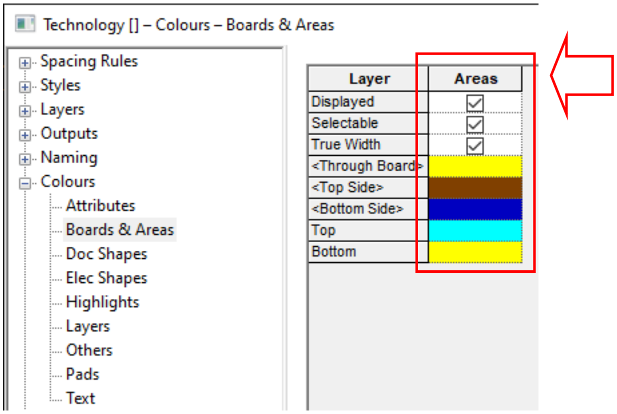
Within the **Footprint Editor**, you can define a footprint as **Defined Mirrored**, which means it is designed as it would appear when mirrored.



If the **Designed Mirrored** box is checked, the orientation of the footprint, as defined in the editor, is the mirrored state. It is likely that you will also want to set the **Mirrored** state as described above. The normal state is to leave this box unchecked.

Area Colour in PCB Doc Symbol Editor

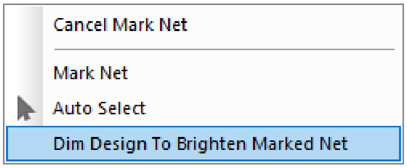
Area colours can now be defined in the **PCB Doc Symbol Editor**.



Background Dimming on Mark Net

In the PCB editor, there is a new functionality that allows you to toggle the background dimming whilst using **Mark Net** (Selected Net) in **Latch Mode**. This toggle is listed in the context menu whilst within this mode.

When in **Mark Net** mode, right click and toggle **Dim Design To Brighten Net** mode.



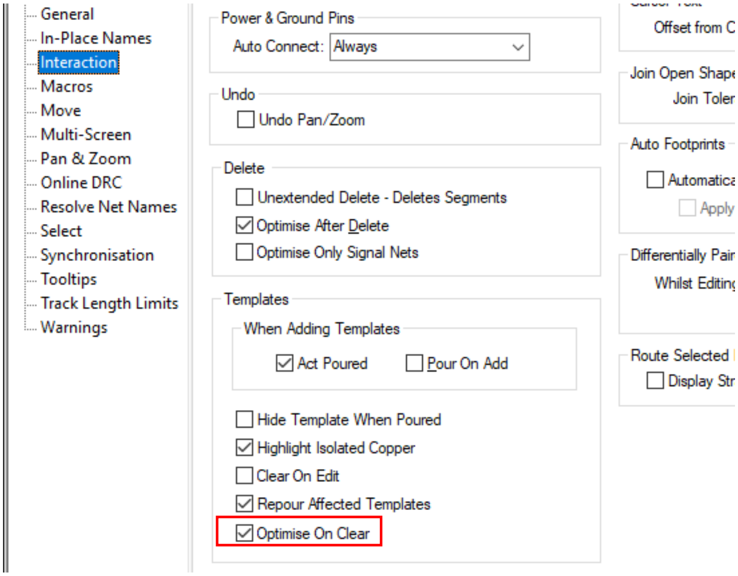
Copper Pour Multi-Threading

The option, **Pour All Templates** can now use multiple threads. For large or complex designs, or designs with many templates, this significantly speeds up the processing.

The selection to **Enable Threads** for **Copper Pour** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Optimise after Clear All Templates Multi-threading

Multiple threads can now be used when **Optimising** nets after **clearing all templates**. The **Optimisation of nets** is performed when Clear All Templates feature is run. **Optimise** is also controlled by **Optimise On Clear** switch on the **Interaction** page in **Options**. For designs with large nets, unchecking this option will significantly speed up the clearing of Templates.



The selection to **Enable Threads** for **Optimise** after **Clear All Templates** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Change Style DRC Checks Multi-Threading

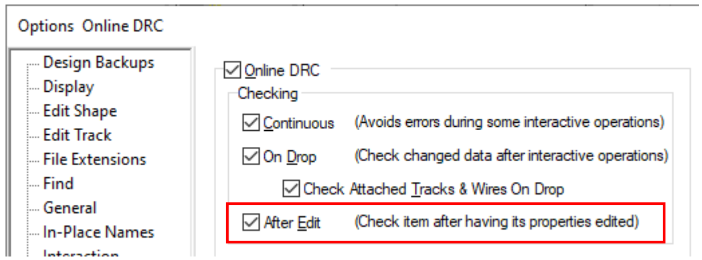
Design Rules Checking can now use multiple threads when after **changing styles** of many items.

The selection to **Enable Threads** for **Design Rules Checking** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Change Style Performance Update

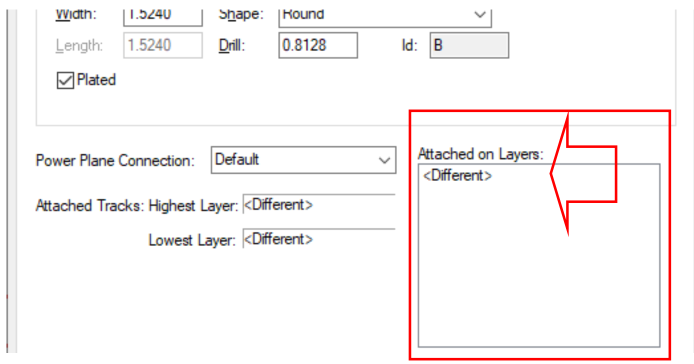
Changing a Style

Changing a Style is now classed as an **After Edit** DRC action. If this option is disabled in the **Options** dialog, **Online DRC** page, then DRC will not be performed after changing style.



Vias Attached On Different Layers

When multiple vias are selected that have different connectivity, the wording <Different> will be seen in the **Attached on layers** list box, this makes opening the **Via Properties** dialog much faster when a lot of vias are selected.



This feature was back-fitted to 10.5

Multi-Threaded Design Rule Checking

Design Rules Checking can now use multiple threads when doing **Spacing checks**.

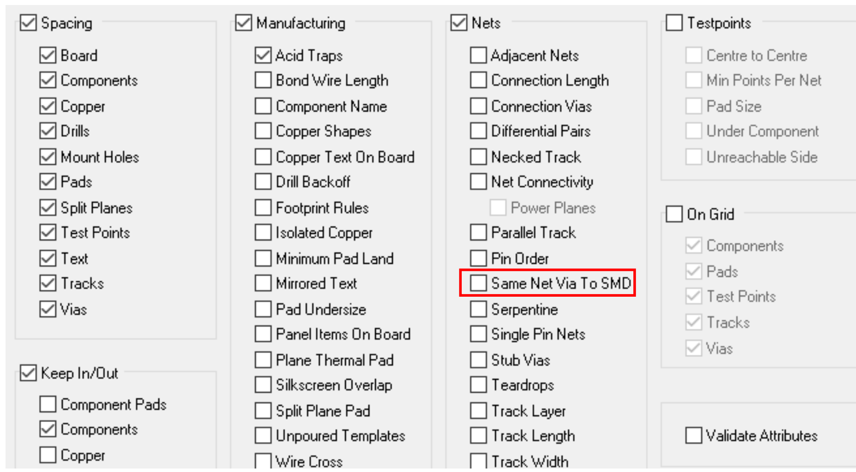
The selection to **Enable Threads** for **Design Rules Checking** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

New Design Rule Checks

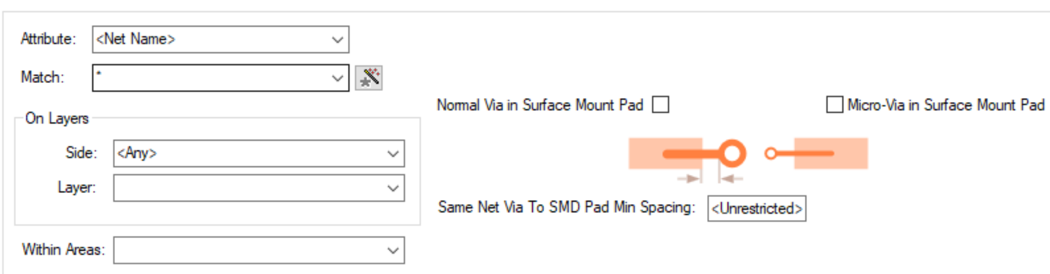
The **DRC** dialogs for PCB, Footprints and Panels have been rearranged for clarity with options under each category are now sorted in alphabetical order. The category **Testpoints** have been split from **Manufacturing** for additional clarity.

Same Net Via To SMD

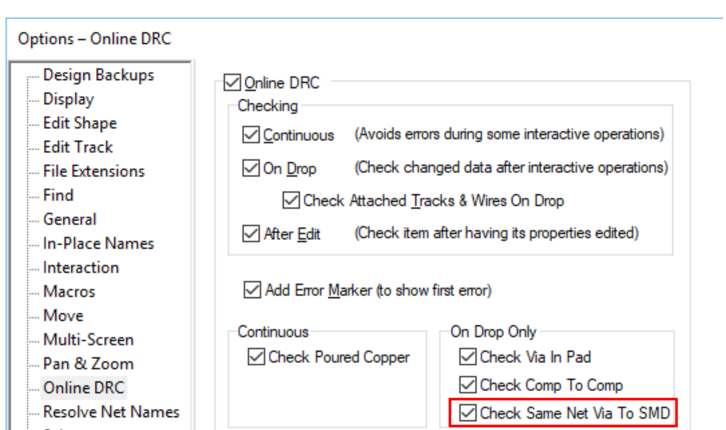
There is a new option in **DRC** to check the spacing between Vias and SMD pads on the same net.



The **Same Net Via To SMD Pad** spacing is defined in your **Technology** dialog under **Via** rules (**Rules – DFM/DFT**).

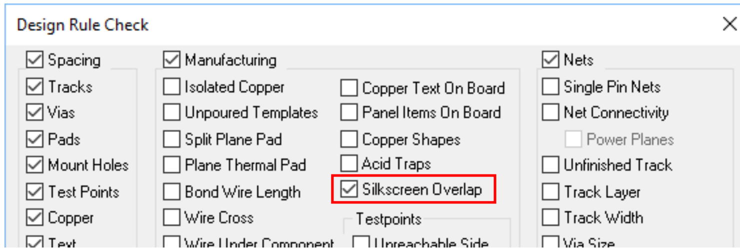


This can also be enabled in **Online DRC** by selecting the **Check Same net Via To SMD** check box in the **Options** dialog and **Online DRC** page. When using Online DRC, the clearances will also be displayed.



Silkscreen Overlap

There is a new option in **DRC** to check for any pads that are overlapped by silkscreen items.



Time To Process shown in DRC Report

The **DRC** report now shows the time taken to perform the check. You'll find this in the summary at the bottom of the report.

```
Total:
     7 Board to Component Error (B-Cm)
     1 Board to Copper Error (B-C)
   223 Board to Pad Error (B-P)
    54 Board to SMD Pad Error (B-SM)
     4 Component to Component Error (Cm-Cm)
    47 SMD Pad to SMD Pad Error (SM-SM)

Checking took 0.657 Seconds
Number of errors found : 336
```

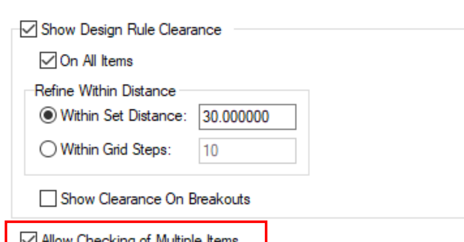
This feature was back-fitted to 10.5

Changes to Online DRC

Display Clearances for Multiple Items

You can now see clearances when moving multiple items, for example, multiple Tracks as well as multiple Components.

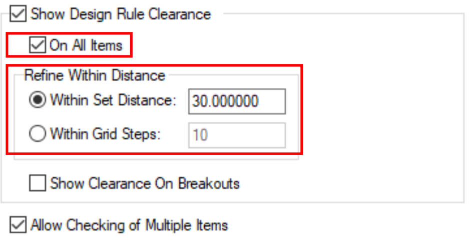
The **Online DRC** dialog in **Options** has been changed to **Allow Checking of Multiple Items**).



Changes to Show Design Rule Clearance

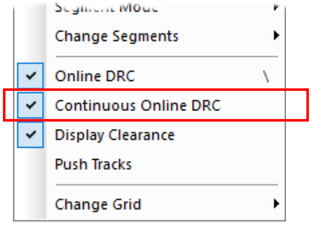
Within the **Online DRC** page of **Options**, display clearances section has been changed. Instead of three radio buttons for **On All Items**, **Within Distance** and **Within Grid Steps**, **On All Items** is now a check box. This allows for refining the clearances when moving a selection that has multiple possible clearances. When checked, the clearances for the selected item will be shown, but when another selected item comes within the defined distance or grid steps of an item it has a different clearance with, the clearance for that item will change.

The **Refine Within Distance** words will toggle between this and **Only Within Distance** depending on the **On All Items** check box selection.



Continuous Online DRC switch added to context menu

A new switch has been added to the context menu to enable **Continuous Online DRC**. This enables you to suspend **Continuous** mode so that errors can be added (whilst using **On Drop** mode) without having to use the **Options** dialog.



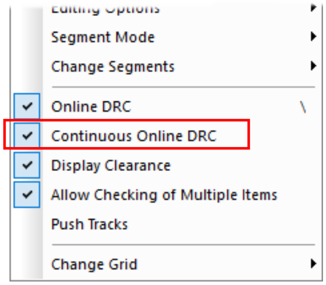
‘End Track On Via’ Clearances

Continuous DRC and Via Errors

Via Spacing clearances are now shown for **Vias** added using the **End Track On Via** option when editing a Track. With a Via on the end of the track and **Continuous DRC** enabled (from the context menu), the clearance shown will toggle to the error colour when the via is in error.

The clearances for the via when using **End Track On Via** will only be shown if the **Allow checking of multiple items** option is enabled (it is no longer called **Allow checking of multiple components**).

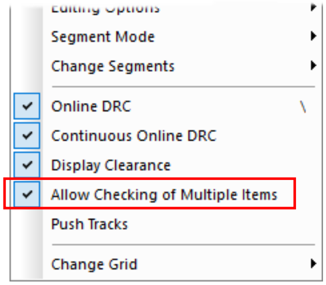
If **Continuous Online DRC** is disabled, the clearances will use the error colour if either the via or the Track breaks the **Spacing Rules** defined in your Technology. If **Continuous Online DRC** is enabled, the Track will be ‘blocked’ from breaking any Spacing Rules, and the via will make clearances the error colour if it is breaking any Spacing rules.



Allow Checking of Multiple Items

Allow Checking of Multiple Items has been added to the context menu when using the **End Track On Via** option. If this option is not checked, only the clearances for the Track will be seen. If it is checked, the clearances between the Track and the item, or the via and the item will be seen; whichever one is closest to being in violation.

If the **Clearances** are being displayed on **all items**, the clearance for the Track will be shown, and the clearances for the via will be shown when the via is within the distance defined in the '**Refine within distance'** section of **Options**, **Online DRC**.

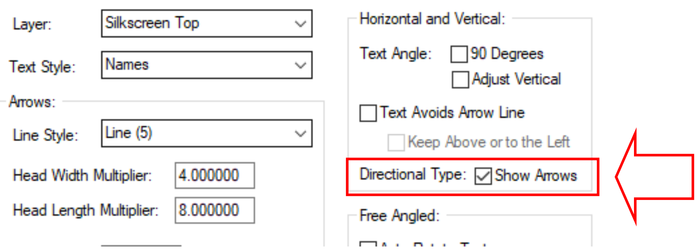


This option works for **End Track On Via**, **End Track On Testpoint Via** and **End Track On Testpoint Pad**.

Dimensions Changes

Hide Arrows on a Directional Dimension

You can now hide the arrows on a Directional Orthogonal Dimension. A new check box, **Directional Type: Show Arrows,** on the **Defaults Dimension** page, allows show or hide arrows.

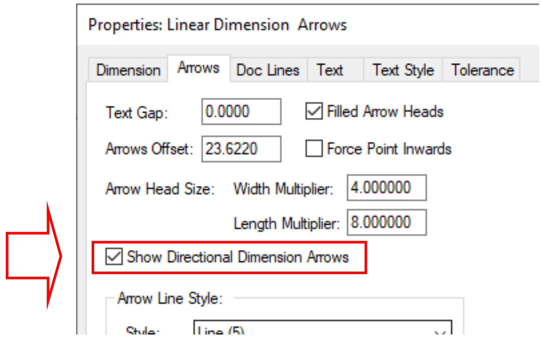


When you add a new directional dimension, the arrows will be shown depending on this setting.

Show Arrows checked

Shows Arrows unchecked

If you select a directional dimension in the design, and use **Properties**, the **Dimension Arrows** page now includes a **Show Directional Dimension Arrows** check box that you can use to show or hide the arrows for the selected dimensions.

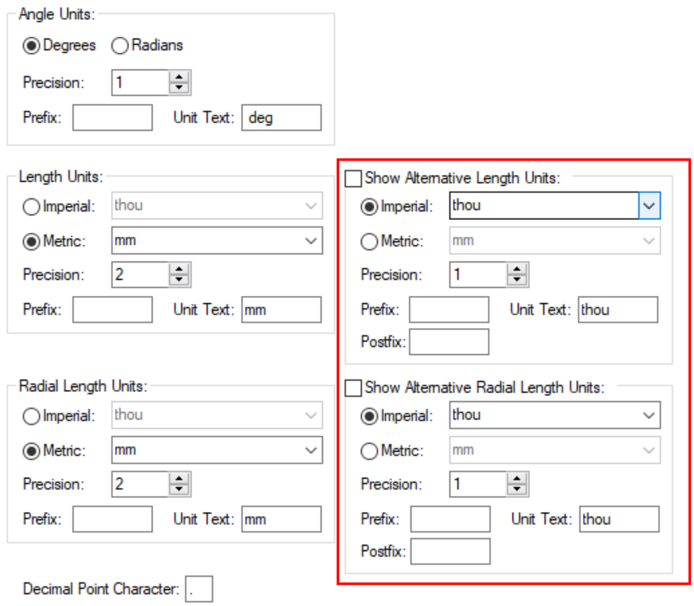


Linear and Radial Dimensions – Show both Metric and Imperial Units

Linear and Radial dimensions can display both Metric and Imperial units for the same dimension.

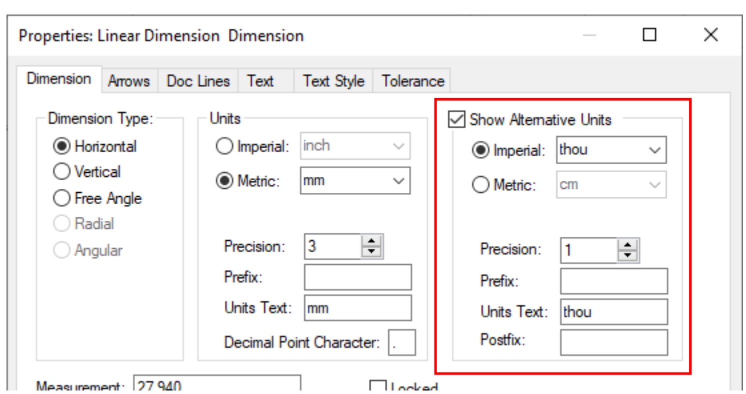
Additional check boxes to **Show Alternative Length Units** and **Show Alternative Radial Length Units** have been added to the **Defaults Dimension Units** page.

When selected, the alternative units text is added after the normal dimension distance text, enclosed in the Prefix and Postfix text, for example **4141 thou (22.61 mm)**. Brackets and formatting can be added using **Prefix** and **Postfix** entries.



If you select a linear or radial dimension in the design, and use **Properties**, the **Dimension** page now includes a **Show Alternative Units** check box that you can use to change the alternative units displayed for the selected dimensions.

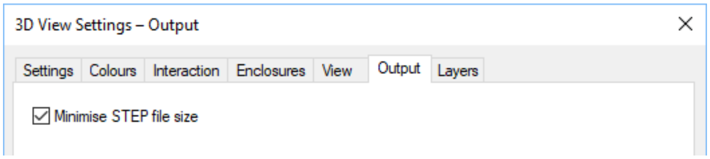
There is a new section to accommodate this:



STEP 3D Changes

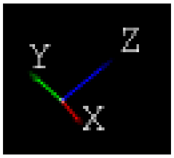
Minimise STEP File Size Option in STEP Output

There is a new option to **Minimise STEP File Size** available on the new **Output** page of the **STEP 3D Settings**. This reduces the size of the generated STEP file by omitting non-essential parametric curve data. This is set on by default.



Improved Trihedron Axes Indicator for STEP Preview & Models

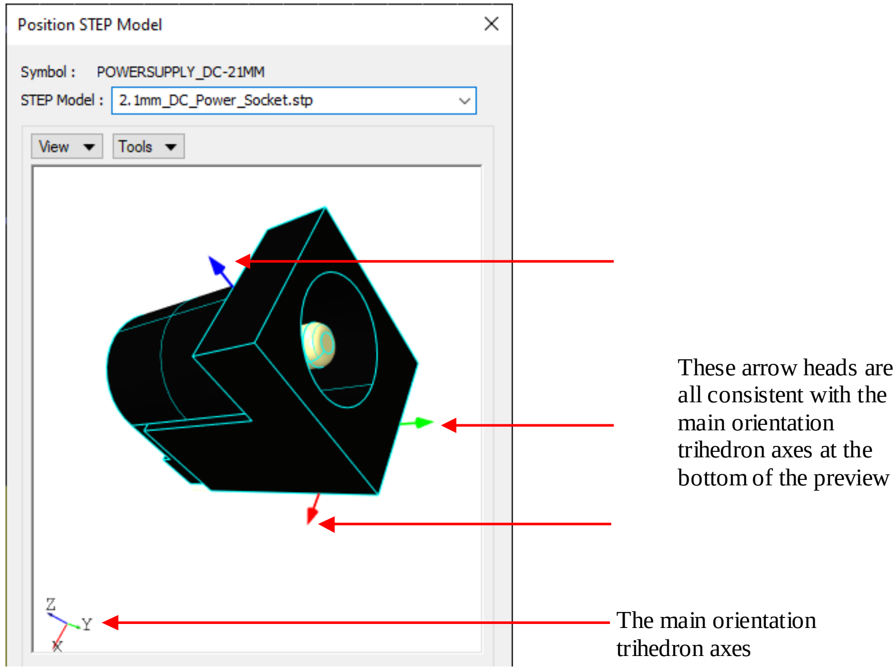
The old axis indicator 'nodules' have been replaced with more standard trihedron axes. This displays arrows designating the positive direction away from the origin on that axis, with the red, green, and blue colours matching the axes.



As well as the main 3D Preview window, the trihedron axes have also been added to **3D View Settings**, **Enclosures** (**New** and **Edit** buttons) and **Library – STEP Models**.

STEP Model Trihedron View

The model trihedron axes (in addition to the overall trihedron axes) is now also displayed in the **Footprint editor** and **3D Settings Enclosures**, **Position STEP Model**. The lines and spheres in previous releases have been replaced with colour-coordinated arrow heads, to represent the direction the model is pointing away from origin.

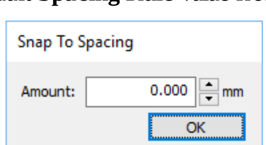


Added ability to 'Align' STEP items

In the **STEP Preview**, a new **Align>** mode can be selected from the **Tools** menu. You can select any two shapes, faces, edges or vertices (or any combination thereof) and the first selected item will instantly snap to meet the second. This allows you to select an item in the 3D Preview and align a sub-selection of that shape with a sub-selection of any other shape, such that they're inline on that axis.

Four new modes are available to select from the **Align>** option: **Snap To**, **Align by X**, **Align by Y** and **Align by Z**

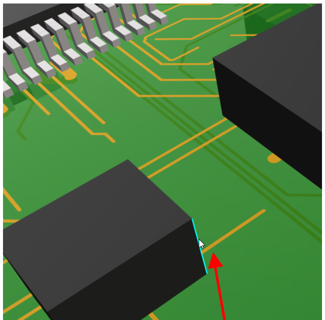
When any of these modes are selected, the Spacing threshold can now be declared prior to aligning on an axis or snapping to, meaning that the two points selected will be exactly that distance apart after the operation. A dialog is presented to choose the **Snap to Spacing** value. The dialog is pre-set with the default **Spacing Rule** value from the **PCB design**.



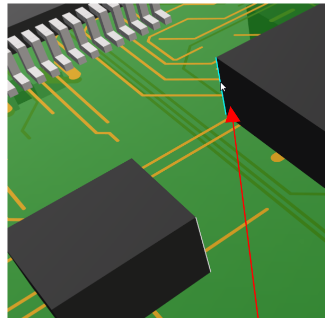
The **Snap To** option is an interactive mode where the first face is selected and the item snapped to the second selected object.

This will move the STEP model (along with the Component), into a new position chosen. This positional change will be reflected in the PCB design with a reposition of the Component.

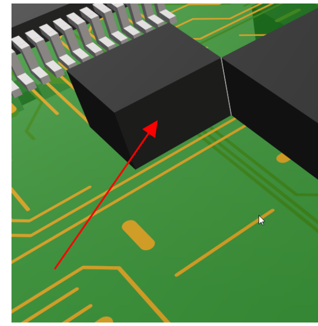
The images below show the process for alignment:



Select the first vertices



Select the second vertices

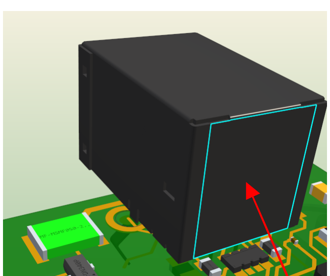


The first shape will move to align

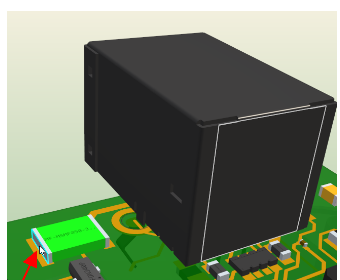
Added ability to 'Orient STEP items

In the **STEP Preview**, with **Orient** mode selected from the **Tools** menu, you can select any two faces of any 3D Items (including two of the same item), and the first face, along with the attached shape, will **rotate** around to face in the same direction as the second face.

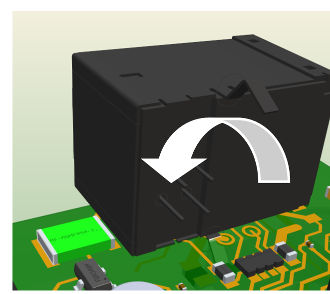
This will move the STEP model (along with the Component), into a new position chosen. This positional change will be reflected in the PCB design with a reposition of the Component.



Select the first vertices or face



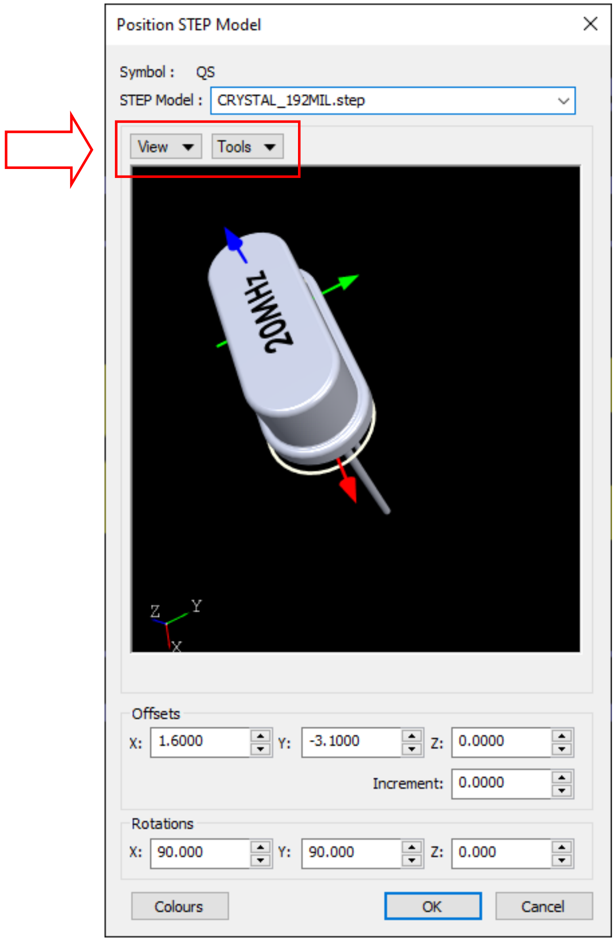
Select the second vertices, the edge of the capacitor in this example



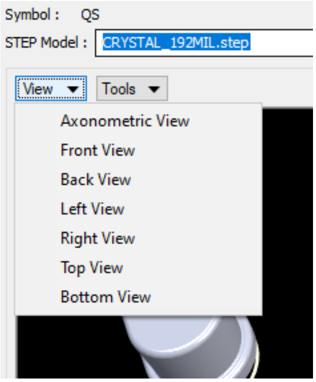
The first shape will rotate to match the capacitor

View and Alignment options added to 'Position STEP Model' dialog

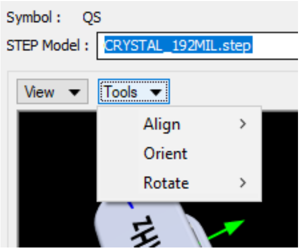
View orientation and **Tools** options have been added to **Position STEP Model** dialog on the **Tools** menu when editing a **Footprint**.



The drop-down **View** menu enables you to select one of the preset views available. These are the same as the views available on the existing **3D View Orientations** toolbar.



The drop-down **Tools** menu will enable you to interactively position and align your STEP model with the footprint. Options for **Align**, **Orient** and **Rotate** are available. These are same as described above for the main **STEP Preview** window.

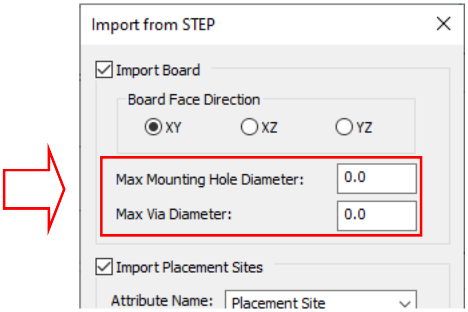


Design Units now used

Another change on this dialog is that all units shown will now be current design units and not forced to mm (Metric) as they were previously.

Import Mounting Holes and Vias into design from STEP Model

The **Import STEP Board** dialog, available from the **File** menu and **Import Design Data on a STEP file**) has been extended to provide additional fields to provide max sizes of **Mounting Holes** and **Vias** seen in the STEP model being imported to the design data. When a circle is discovered that is smaller than the specified mounting hole diameter, the circular element will be removed from the shape and a mounting hole will be created in its place, or a via if it is small enough for that. It assumes that mounting holes are bigger than via holes so it does the tests that way round.

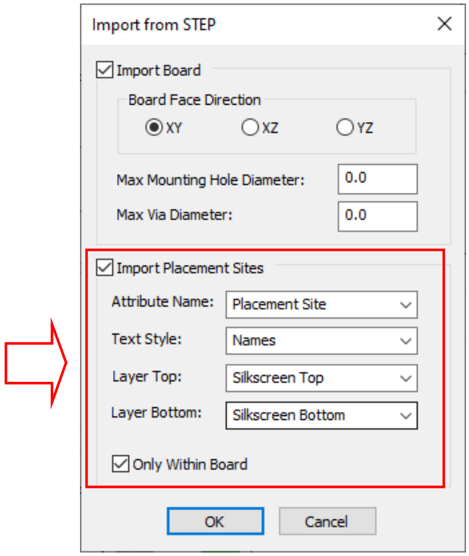


Import STEP Board Placement Sites

The **Import STEP Board** dialog, available from the **File** menu and **Import Design Data on a STEP file**) has been extended to provide additional options to allow the location of items in the STEP data to be turned into **Placement Site attribute positions** in the design. These will only be added if they are inside a board outline in the design. The STEP files have an object location catalog for the top-most elements, components like connectors, switches and other objects that the mechanical department preplaces on the board for placemen of those objects.

Additional controls allow you to choose the **Attribute Name** to use, the **Text Style**, and the **Top** and **Bottom Layers** for the attribute positions that are added to mark placement sites. The Top layer is used for STEP file locations where Z >= 0, and the Bottom layer for Z < 0.

Choose from the **Attribute Names** listed in the drop-down list, or enter a new attribute name.



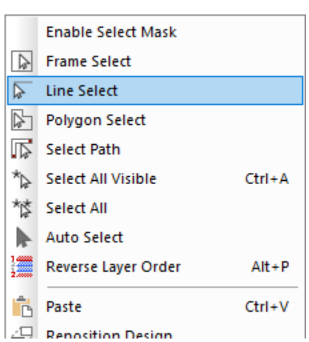
You can also use the **Only Within Board** check box to choose whether or not to keep any positions that are outside all board outlines.

3D Package Viewer Retirement

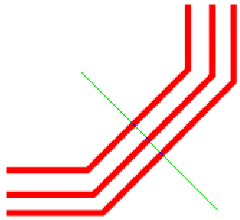
The old **3D package viewer** (not associated with viewing STEP models) is no longer available on the menu. This has been retired and removed along with **Packages** feature from the **Library Manager** and **3D Settings** options from the **Setup** menu. If you wish to continue using this feature, you can add it to your menus using the **Customise** option or use the **Run Command** option or assign the command to a shortcut key.

Line Select Mode

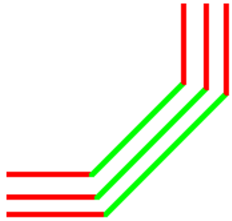
With nothing selected in the design, right click and the new **Line Select** selection mechanism will be available, if something is already selected it will be on the Select sub folder. Use this to define a straight line to select only items that cross that line.



Once selected from the menu, draw the line across the items you wish to select.



The selection will then be made:

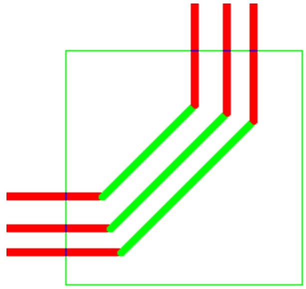


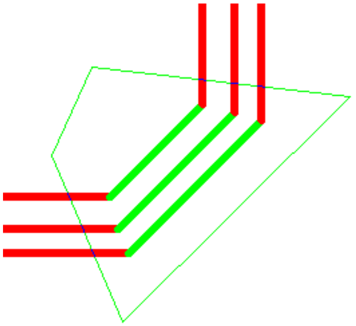
Like **Frame Select**, keep the **Ctrl** key pressed down to alter the selection. Use the **Shift** key to extend the selection to whole items.

Line Select is also available from the context menu in **Align** mode and **Latched** operations such as Move when nothing is initially selected.

Changes to Frame Select and Polygon Select Modes

Both **Frame Select** and **Polygon Select** modes will now act on individual track segments to make segment picking more versatile. This will operate with the option in **Options**, **Select** page, **Select if Completely Framed** is set to on or off.





Changes to Double Click to Edit Mitre

Double click on a mitre has been changed so that it will enter the **Edit Mitre** mode regardless of its mitre size. The **Edit Mitre** mode is also available on the **context** menu for a selected mitre.

For adjusting multiple mitred corners, such as parallel tracking, this change works in conjunction with the three selection modes described above (Line, Frame and Polygon).

In Version 10.5, it was changed to work on double-click if it was less than twice the default mitre size as defined in the Options dialog and Edit Track option. This change has been reversed in Version 11.

Insert PCB Track more responsive

Snapping to a line orthogonal with the target (item at end of connection, or point between the two connections if a Differential Pair) is now responsive and easier to snap to, retaining the pad centre to the orthogonal track.

You can now also avoid snapping to the Pad centre by holding the down the **Shift** key whilst moving the track.

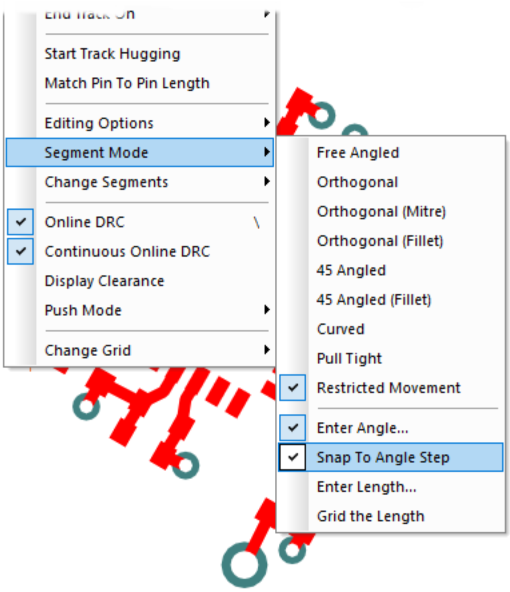
This feature was back-fitted to 10.5

Restricted Movement Segment Mode – Snap To Angle Step

When editing a Track or shape in **Restricted Movement Segment** mode and the restricted angle is not zero, you can now change how the angle is snapped to.

Once routing in **Restricted Movement Segment** mode, from the **Segment Mode** context menu, there is a new option **Snap To Angle Step**.

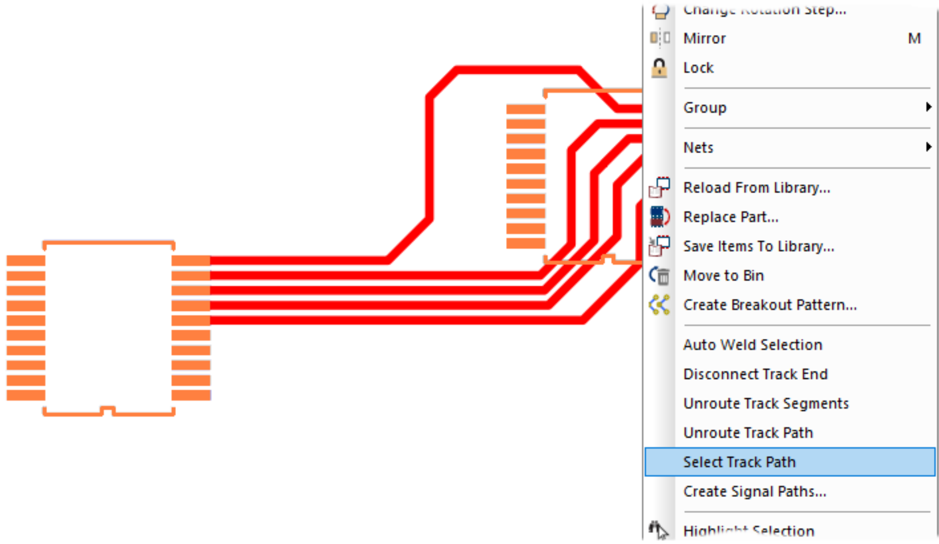
This option will only appear if the **Snap Angle** is a multiple of the restricted movement angle. Unselect this option to snap to any 45-degree step relative to the restricted movement angle (as it worked previously). Using this new mode, you can snap to any multiple of 30 degrees. Previously, it would only snap to the angle entered and any 45-degree step from that angle (30, 75, 120 ...).



This feature was back-fitted to 10.5.

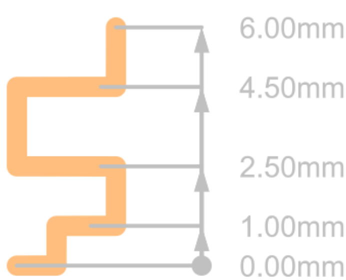
Select Track Paths by selecting Components

In **Select Mode**, if 2 or more components are selected, if you right click, the context menu displays **Select Track Path**. Selecting this, the current selection will be deselected and all track paths directly between the previously selected components will be selected.



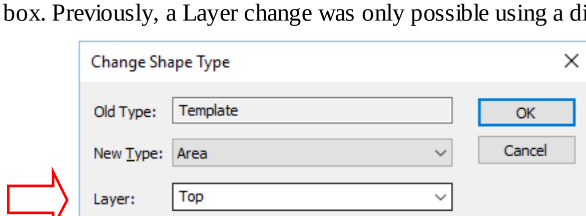
Copy/Paste into new Shape Type

You can now **Copy** and **Paste** a single track as a generic Doc Shape. This feature can be accessed when using **Insert Doc Shape** and then using **Paste** before starting to draw a shape. The selected, and copied track shape will be pasted in as a doc shape.



Layer Control Added To Change Shape Type

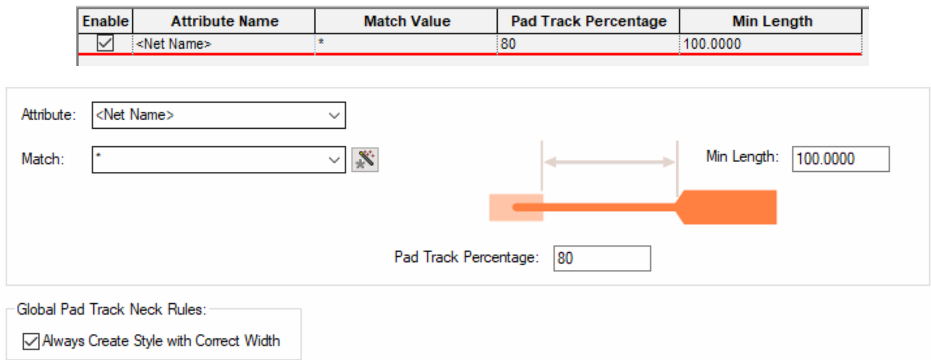
You can now change the **Layer** in the **Change Shape Type** dialog using the **Layers** drop down list box. Previously, a Layer change was only possible using a different option or dialog



Pad Auto Necking

A new rule page has been added to **Technology DFM/DFT rules** called **Pad Track Neck**. This specifically for Pad necking into and out of pads.

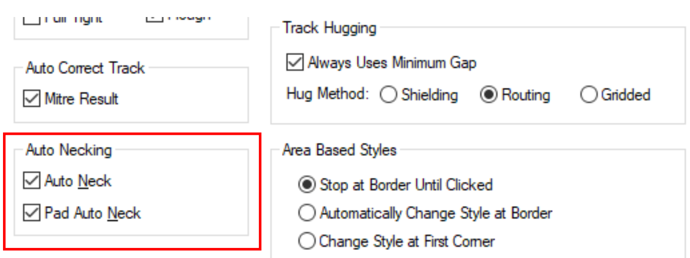
This rule allows you to define a **Pad Track Percentage** which is the percent that the necked track width will be in comparison to the pad width (for example, if the pad width is 50 and the percentage is 80% then the necked tracks width will be 40).



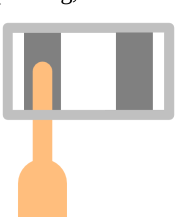
The **Min Length** option defines the length of the necked segment. This length is measured from the edge of the pad to the tip of the '**fat**' track. The completion shape also takes this new calculation of the length into account when creating a necked completion shape.

The **Global Pad Track Neck Rules** at the bottom section includes the option to **Always Create Style With Correct Width**. With this option disabled, when the necked track is created and if the style is not already in the design, a message will display asking if you want to add the style. If the option is enabled, then the style will be added without warning.

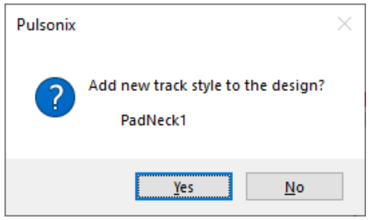
For this rule to be operative, you must have the option from the **Options** dialog, **Edit Track** page and **Auto Neck** enabled. It can also be enabled in the editing options section of the context menu when editing a PCB track.



When operating, it will look like this when you click to make a corner:



If the check box to **Always Create Style With Correct Width** is enabled and a new Track Style needs to be added, a warning dialog is presented:



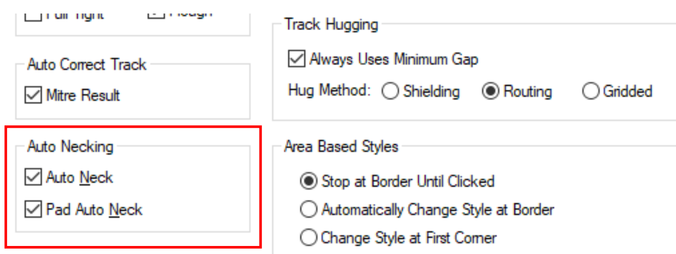
The new Track Style will be created at the percentage width to that of the pad it is exiting.

Auto Necking

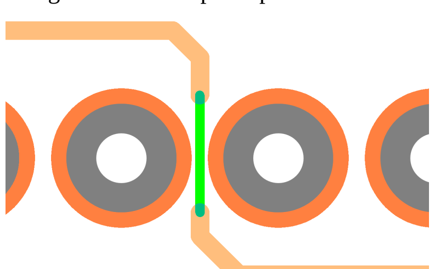
You can now neck to and from pads or through gaps that would normally cause a DRC error. It can be used when routing anywhere in the design and will neck down the current segment to avoid DRC errors. Once avoided, it then reverts to the regular style after passing the items that would cause the error, you can then continue routing.

This mode can be used by either clicking directly on the pad or using the completion shape. When using the completion shape for pad auto necking, it will show the necked segment at the end of the completion shape.

You can enable this option from the **Options** dialog, **Edit Track** and by selecting **Auto Neck**. It can also be enabled in the editing options section of the context menu when editing a PCB track.

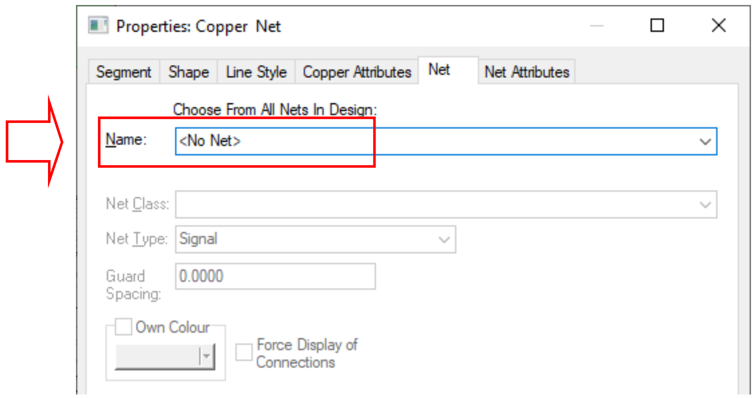


When editing a track near a pad a potential violation takes place, the necking will be performed:



Add Item to Net in Net Properties Dialog for Copper and Templates

You can now use **Properties** to add an item to a net, for example a Copper shape or Template.



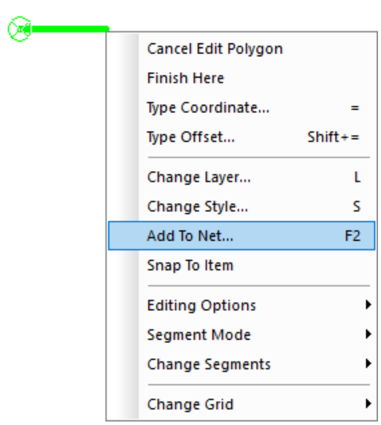
An item not on a net will show as <No Net>. Changing this to a net name will add it to the selected net. Clearing the net name will add it to a default net, but it will ask you first.

If multiple items are on different nets it will now show as <Different> (whereas, it used to be blank). By typing a single net name will merge the nets to one net (after asking about merge).

If multiple nets or multiple no nets, and the net name is cleared, after asking if you want default net names, you will be asked if you want to merge all nets into one default net, or to create multiple nets. (Note: In Schematics, you will not be asked and individual default nets will always be created).

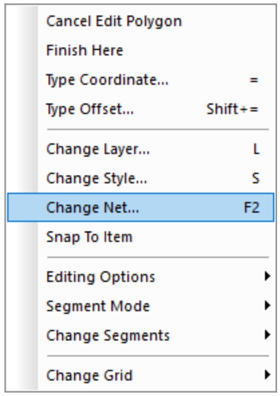
Add Copper & Template Shapes to Net on Insert

Whilst adding **Copper** or **Template** shapes to your design, you can now add them to a net using the **Add To Net** option from the context menu option.



This can be done either before you start the first point, or whilst adding the shape. If you have a similar item preselected the net will be taken from it (previously, this worked for templates), but it has been changed so that if you preselect a different type of item (a track for example) the net will be taken from that item and a warning will be issued to ask if you want to use that net.

If whilst adding the shape you wish to change the net name or remove it from the net, use the **Change Net** option from the context menu and edit the name or remove it in the dialog.

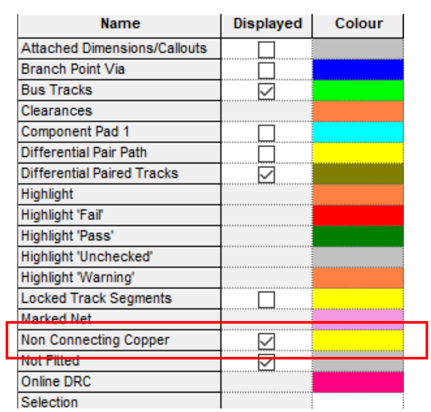


This feature was back-fitted to 10.5.

Non-Connecting Copper Highlight Colour

A new category for **Non-Connecting Copper** has been added in **PCB Colours** under **Highlights**.

The **Non Connecting** switch on **Technology**, **Copper Pour Rules** and **Copper Shape Properties**, causes isolated copper areas to be marked as Non Connecting. This means that it can remain isolated and does not form part of the connectivity, although it does still retain the same net name. You may require this to maximise copper on the board.

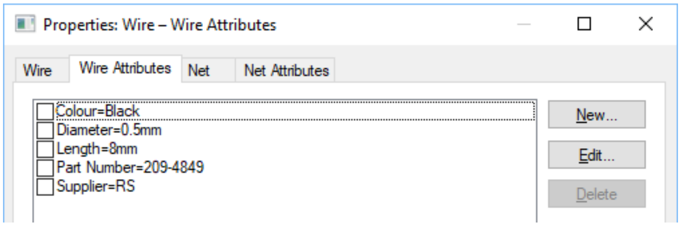


Reversed View Status Saved

When **Reversed View** is enabled in your design, this status is now saved when **Save** or **Save As** are used, this means the design will remain reversed when re-opened.

Attributes on Wires

You can now add attributes to **Wires** in the design using **Properties** or **Add Attribute**. These can be used to hold copper wire material or the wire diameter for example.



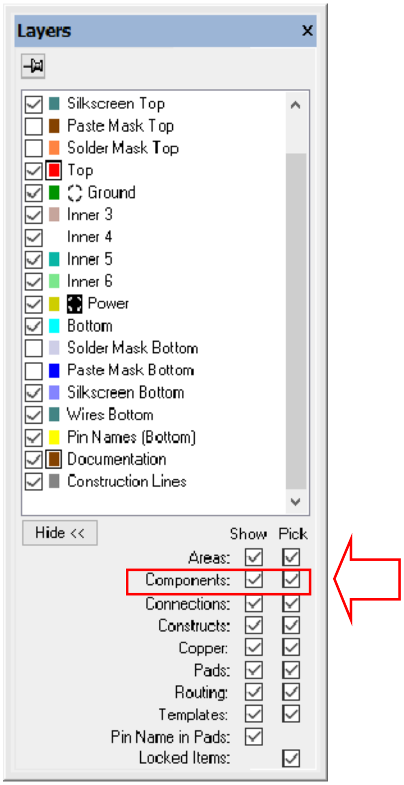
These attributes can then be used in the **Find** bar to select all wires of the same diameter for example.

You can also use the **Report Maker** option to report wire links and their attributes in a bill of materials, along with the standard Wire attributes that already exist (such as Length and Insulated status).

This feature was back-fitted to 10.5

Layers Bar – Components Filter

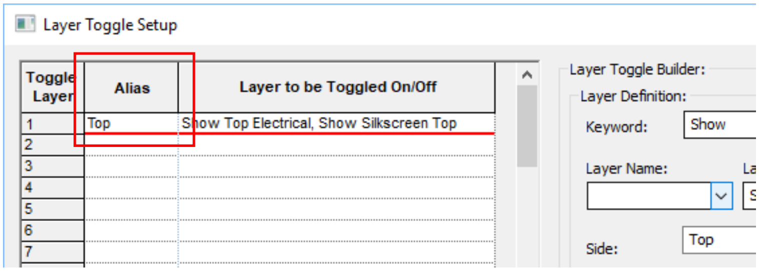
From within the **Filter** section of the **Layers Bar**, you can now toggle the **Show** and **Pick** status of **Components**:



Toggle Layers Changes

Alias Assignments

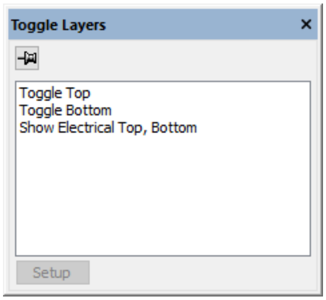
Layer Toggle commands can now have an **Alias** name assigned; this allows for more readable/shorthand descriptions of the command. These names can be defined in the **Layer Toggle Setup** dialog.



Layer Toggle Alisa names can be viewed in the new **Toggle Layer Bar** (*see below*).

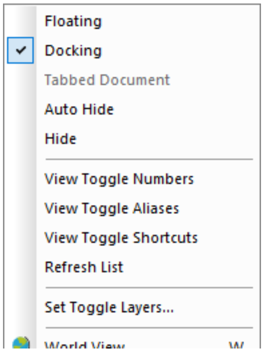
New Toggle Layers Bar

A new dockable **Toggle Layers Bar** has been added. This allows you to activate any of your defined layer toggles on the fly, while in design.



The **Setup** button will take you to the **Setup Toggle Layers** dialog from where you can define the Toggle Layers.

If you right click on the Toggle Layers Bar, a context menu is available:



Three active settings enable you to display additional information if defined in the Toggle Layers dialog on the Toggle Layers Bar: **View Toggle Numbers**, **View Toggle Aliases** and **View Toggle Shortcuts**. The **Set Toggle Layers** switch will take you to where these can be defined.

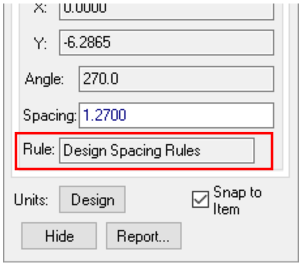
New Toggle Plane Command

A new command has been added to toggle the display of **Powerplanes** on and off.

If you have a plane layer called **Ground** then you would create a toggle **plane ground**, this will toggle the Powerplane display state of that layer in the same way that you get when clicking on the little thermal icon on the layer in the **Layers Bar**.

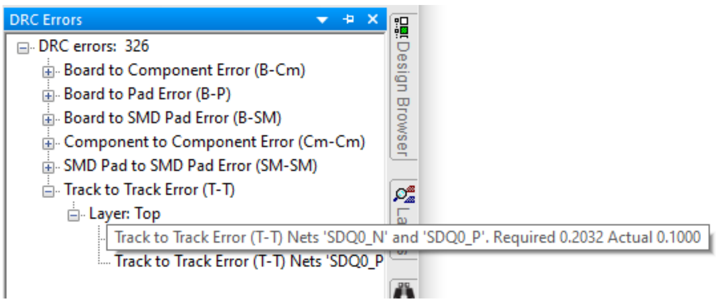
Measure Bar showing Spacing Rule

The **Measure Bar** now displays the rule that corresponds to the calculated spacing.



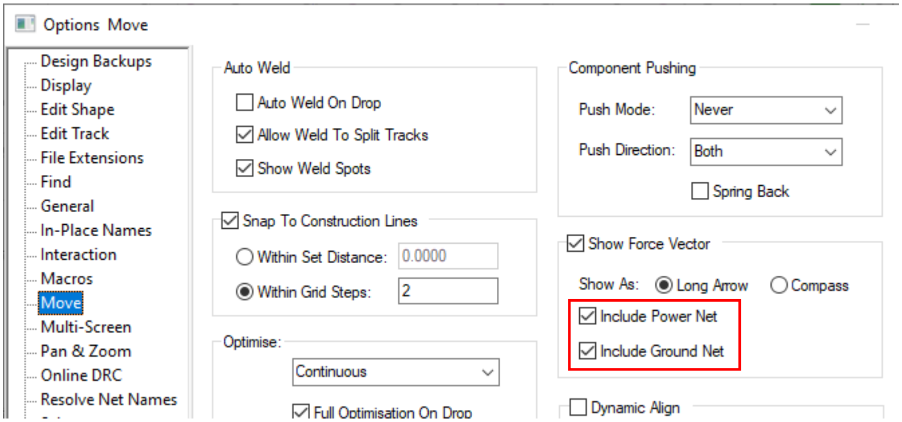
Error Bar - Tooltip available on Error Rule

When an error marker is displayed in the **Errors Bar**, the rule violated can be displayed in a tooltip by hovering the mouse over the marker. This is available when the full error name cannot be displayed.



Placement Vector - Include Power/Ground Net

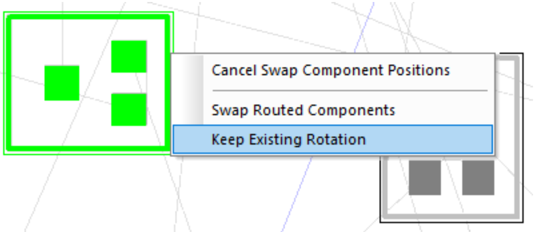
New check boxes in **Options** and **Move** allows you to select whether **Power** and/or **Ground Nets** are to be included in the calculation of the most optimal point for Component placement using the placement **Force Vector**. Power and Ground nets are defined by their **Net Type** (Power or Ground).



Unchecking either of these boxes will change the dynamics of the force Vector. Unless you specifically require these nets to be included on the calculation, we suggest you deselect these.

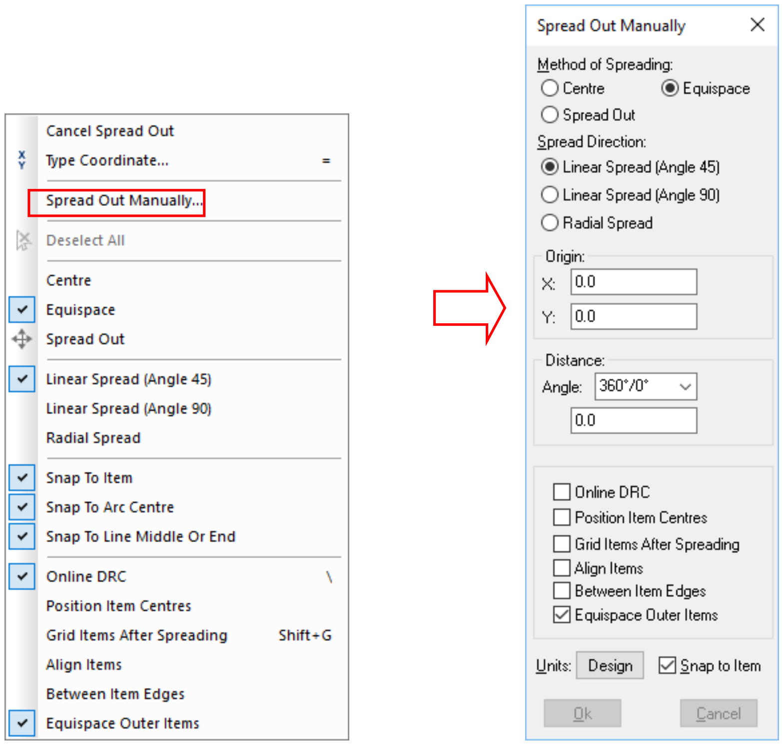
Keep Existing Rotation in Swap Component Positions

In the **Utilities** menu, **Swap Component Positions** tool, there is a new feature on the context menu available during use of this option. You can now **Keep Existing Rotation** if enabled, this will allow swapped components to keep their original rotations but still have their positions changed. If disabled, swap component positions will be completed as before (rotations will be changed to match swapped component).



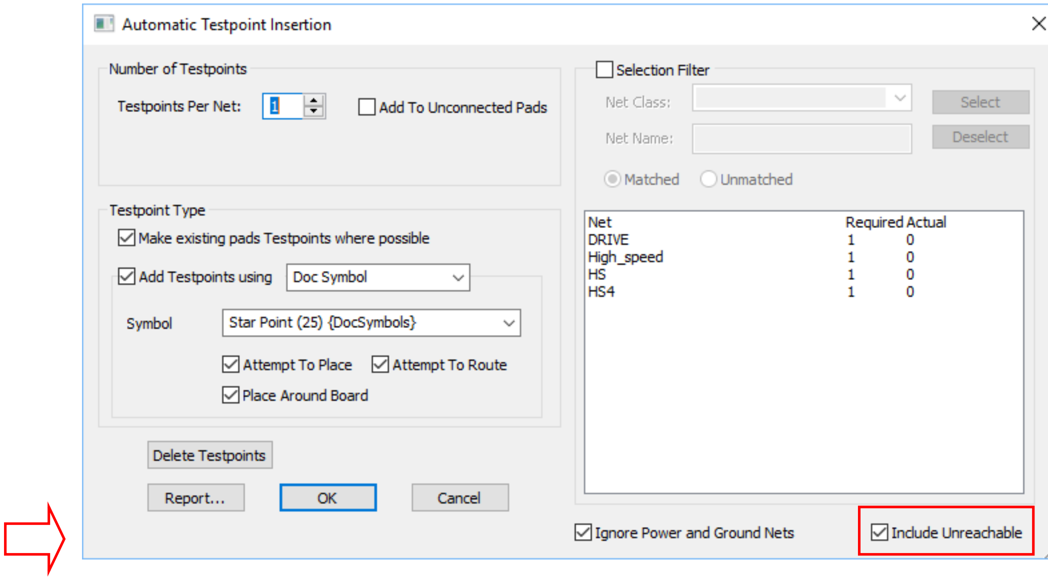
Spread-Out Option Changes

The **Spread Out** utility, available from the **Utilities** menu, now has a companion **Spread Out Manually** dialog, which is accessible from the context menu. This allows for more specific changes to the dimensions and spread distance, plus refinements to spread parameters.



Auto Insert Testpoints – Include Unreachable Testpoints

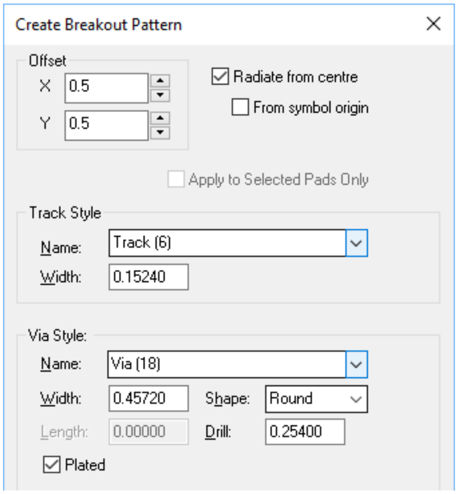
From the **Tools** menu and **Auto Insert Testpoints** dialog, there is now an option to include previously placed unreachable testpoints.



Excluding these would mean that previously placed testpoints, which are unreachable according to the current rules, would not be included in the count, so additional testpoints would be required.

Create Breakout Pattern (BGA Fanout) in PCB

Within the PCB design, there is now a command available in the **Tools** menu called **Create Breakout Pattern.** This is used to create breakout (fanout) patterns within the design without the need to create a Breakout pattern in the footprint first. To use the option, select a component or a pad on the component and use the **Create Breakout Pattern** option on the context menu. A dialog is displayed from which to choose the pattern required:



The dialog will only create a pattern for components that have at least one pad that is on a net and does not already have a routed connection. Applying a pattern will not delete or edit any existing tracks that are attached to pads on the component (no changes will be made to these pads).

When applying a pattern, a DRC check will be performed. Selecting **No** will remove the inserted pattern.

The command for this option has changed from **Create footprint breakout pattern** to **Create breakout pattern**. You will need to reassign this if you have already assigned it to a shortcut key.

Another small change is that you are no longer able to use this option if the selected component in your PCB design already has breakouts.

CAM Plot Changes

Multi-Threading of Gerber Plots

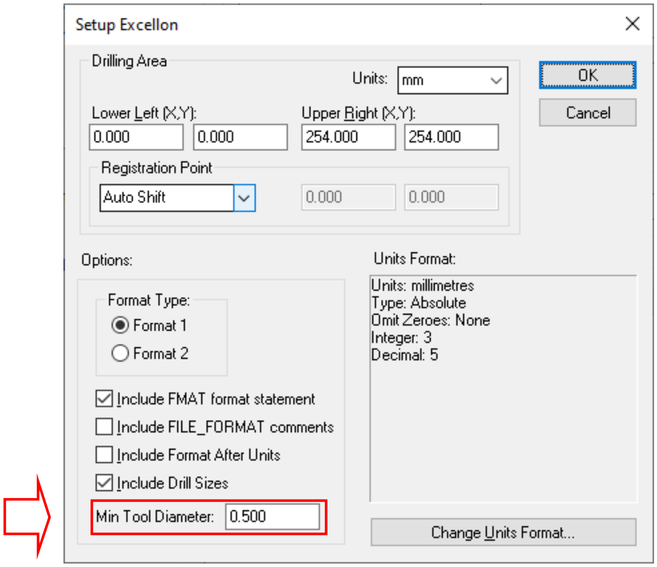
Gerber Plots can now be completed using multiple threads (including Gerber verification plots) when running **CAM Plots**.

The selection to **Enable Threads** for **CAM Plots Gerber export** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Excellon Setup Min Drill Diameter

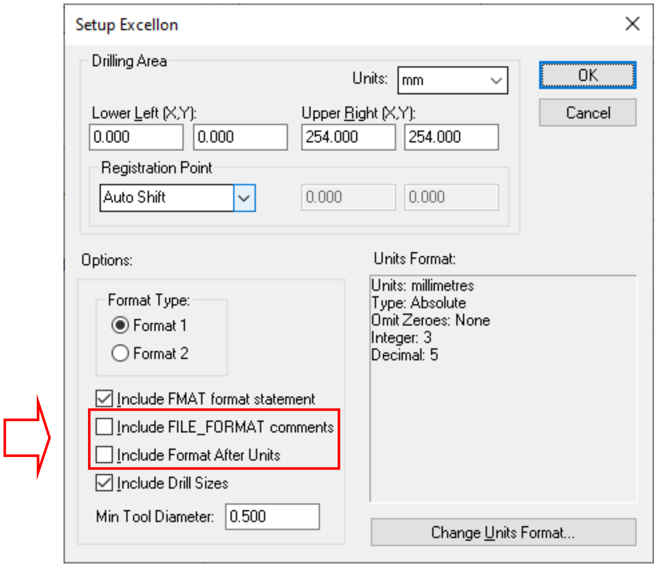
In the **CAM Plot** dialog, **Excellon Setup** page, there is a new setting to define the **Min Tool Diameter**.

This is very useful for the output of **Pad Slots** that have special or intricate shapes. This option defines the smallest valid tool diameter that will be used for these shapes. These would be shapes that are not regular slots that can be milled using a standard drill. When used to route custom shapes, no attempt is made to shrink these shapes so the drill should be as small as possible and in accordance with what your manufacturer can mill.



Optional Formatting information for Excellon output

In the **CAM Plot** dialog, **Excellon Setup** page, you can include optional formatting information at the start of the Excellon file.



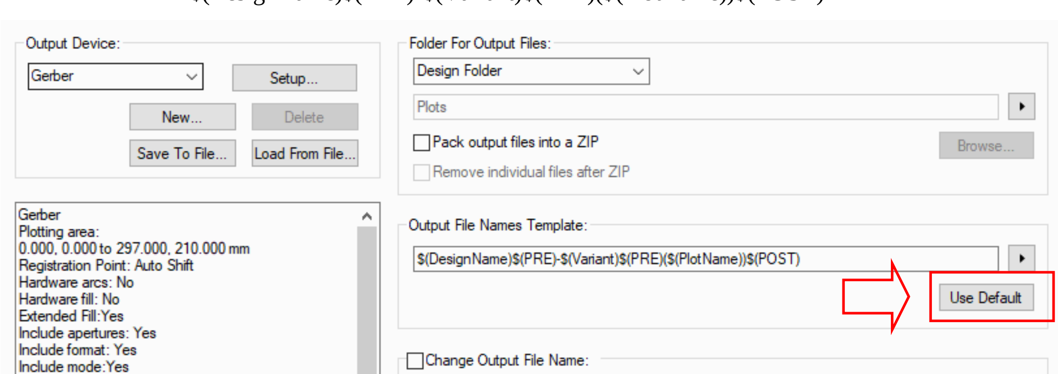
Include FILE_FORMAT comments includes some comments, at the start of the file, of the form **;FILE_FORMAT=2:5** (where 2:5 is the integer:decimal places format), **,INC** is you are using incremental, and **,LZ** is you are using leading zeros. You also get a comment of the form **;TYPE=PLATED** or **;TYPE=UNPLATED** when appropriate.

Include Format after Units includes, LZ (for leading zeros) or TZ (for trailing zeros) followed by the number format shown as 00.00000 for 2.5 (e.g. **METRIC,LZ,00.00000**). This is only relevant if you are using leading or trailing zeros.

Output file names template – Use Default button

A new button, **Use Default**, has been added to the **Output File Names Template** section of the **Plot Settings** tab in the **CAM Plot** dialog. This returns the template to its default value which is:

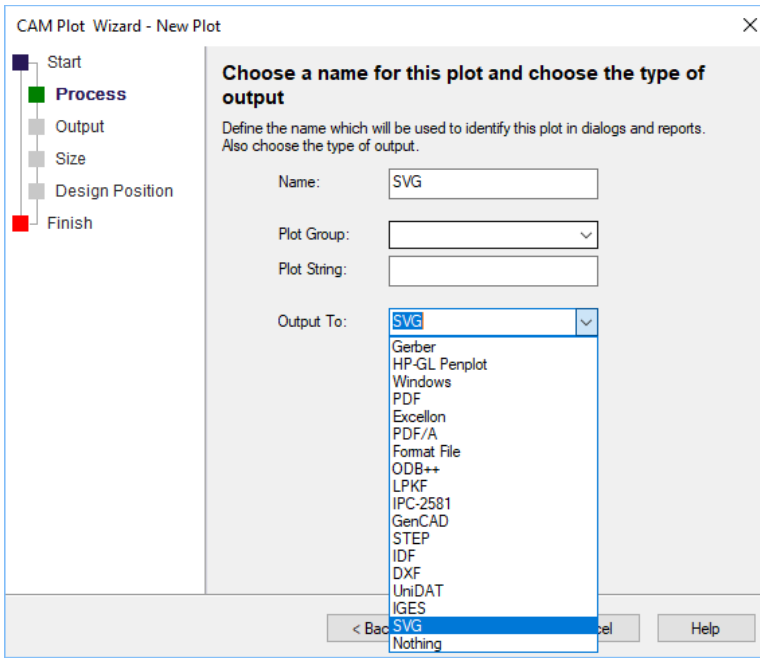
$(DesignName)$(PRE)-$(Variant)$(PRE)($(PlotName))$(POST)



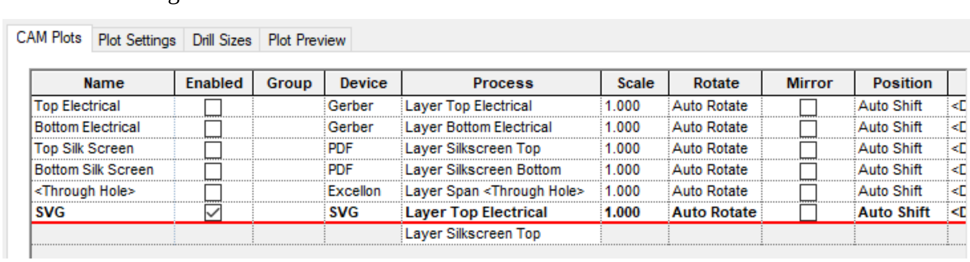
Output to SVG Device

Using the **Output** menu and **CAM Plot** dialog, you can now output an **SVG file**. The SVG file will be an image of the design using the standard rules selected for content and scale. This is available for both Schematic and PCB designs.

The SVG output is available on the list of ‘devices’ on the drop-down list:



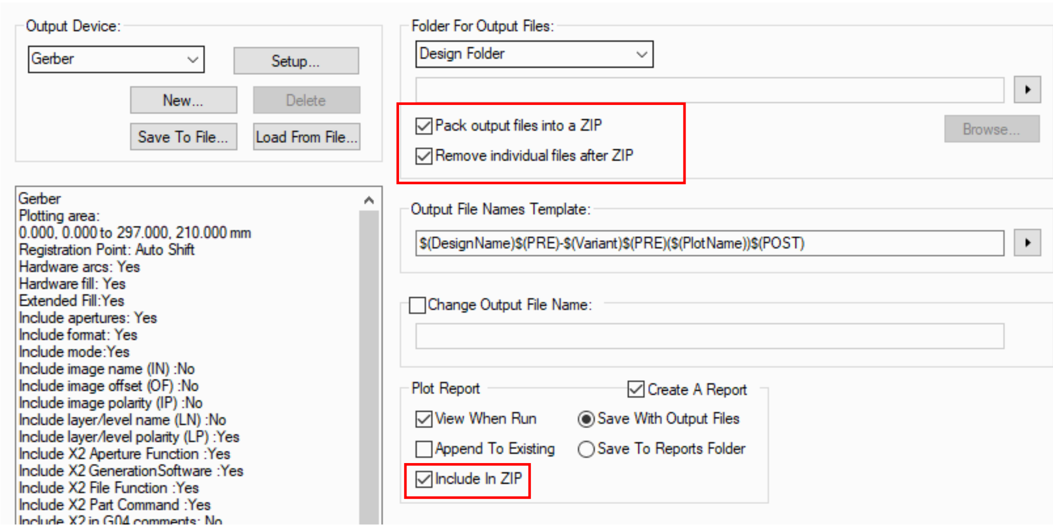
The **CAM Plots** grid shows **SVG** as the **Device** name:



Output to ZIP File

You can now pack all the generated output files (from the **CAM Plot** option) into a **ZIP** file.

From the **CAM Plots** dialog, on the **Plot Settings** page, there are three new check boxes:



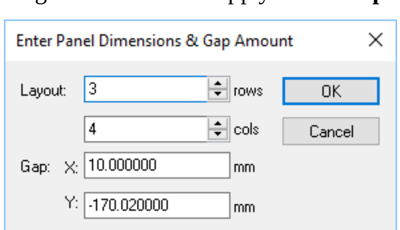
Check **Pack output files into a ZIP** to create a ZIP of all the generated output files in the folder specified in the **Folder for output files** section.

Check **Remove individual files after ZIP** to delete all the individual output files created, leaving you with the ZIP file only.

In the **Plot Report** section, check **Include in ZIP** under the **Plot Report** section to also add the CAM Plot report to the ZIP file.

Panel Editor – Reset Layout Command

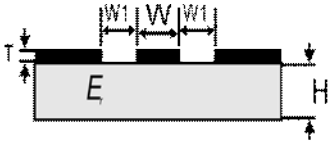
When in the **Panel Editor**, there is a new command **Reset Layout**... This is available on the context menu dialog and is used to re-apply **Panel Gap/Spacing** between PCBs.



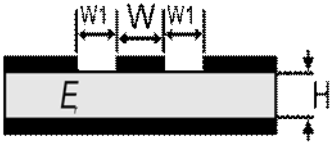
Track Impedance Calculator

Differential Pair (CPWG) impedance calculations have been added to the **Design Calculators**, the CPW and CPWG options also allow for calculation of the conductor space.

Coplanar Waveguide



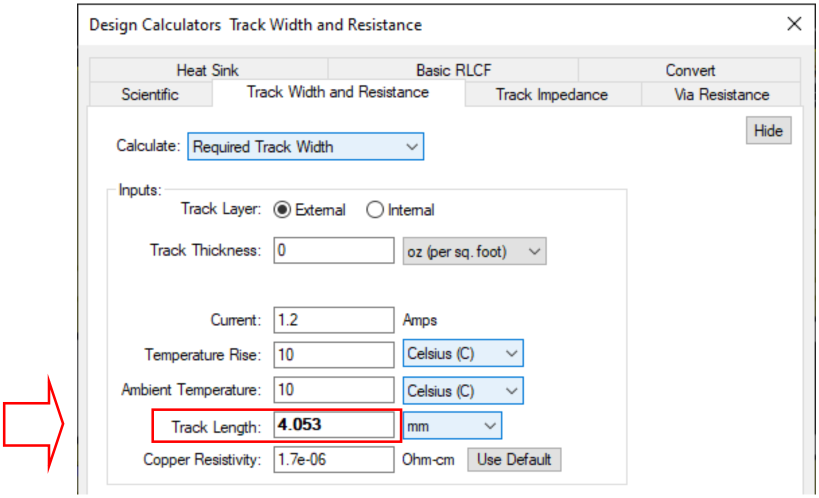
Coplanar Waveguide with Ground



Edge Coupled Coplanar Waveguide with ground

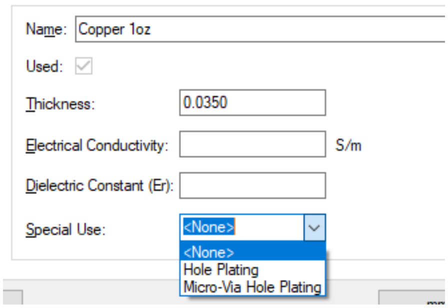
Design Calculators – Values taken from design

The **Design Calculator** dialogs will now take more parameters from the selected track or via. The dialogs will indicate which values have come from the design by using **bold** text.



New Material Parameters
New parameters for **Electrical Conductivity** and **Dielectric Constant** defined in the **Materials** dialog are used by calculators.

Also added is the ability to specify a **Material** for special use **Hole Plating** and **Micro-Via Hole Plating**. This is selected in the **Materials** dialog:



Track Impedance Calculator – Calculating Impedance
Track Geometry
In **on-the-fly** mode, you can use the context menu to choose the **Geometry** to use, or choose **Auto Detect Track Geometry** to try and choose the correct geometry for the track that has been picked (by looking at layers stack and if it is buried in a plane).

The Differential Pair check box will be checked (or not) depending on if the picked track is a paired track.

Relative Permittivity
Will be taken from the **Dielectric Constant** value on the **Material** used by the dielectric layer above or below the layer of the picked track (if it is defined).

Dielectric Thickness and Track To Plane Gap
These will be taken from the **thickness** of the **dielectric layer** above and below the layer of the picked track (if it is defined).

Track Spacing
When picking a Differential Paired track, the **Differential Pair Gap** will be used for the track spacing.

Conductor Space
When picking a track, the **Track to Copper Spacing** for that track and at that point will be used for the conductor space for Coplanar Waveguide calculations.

Track Thickness
Will be taken from the **thickness** of the **layer** of the picked track (if it is defined).

Track Impedance Calculator – Calculating Track Width
Differential Pairs
The Impedance value provided as an input will be the **Differential Impedance** when calculating the required track width for a Differential Pair. The last field in the result will be the **Characteristic Impedance**.

Take Impedance Values from design
The **Characteristic Impedance** will be taken from the **Track Impedance Rules** for the net of the picked track, or to get the Differential Impedance from a Differential Pair (or one of the nets if not on the pair),

Track Width & Resistance Calculator
Copper Resistivity
This will be calculated from the **Electrical Conductivity** value on the **Material** used by the layer of the picked track (if it is defined).

Via Resistance Calculator
Via Depth
This will be calculated from the **thickness** of the **layers** that the via passes through (if they are defined).

Plating Thickness
This will be taken from the **thickness** of the **Material** that has special use **Hole Plating** or **Micro-Via Hole Plating**, dependent on the type of via selected. (if it is defined).

Resistivity
This will be calculated from the **Electrical Conductivity** value on the **Material** that has special use **Hole Plating** or **Micro-Via Hole Plating**, dependent on the type of via selected. (if it is defined).

Interactive High Speed Option – New Rules & Changes

Back Drilling

The **Back Drilling** functionality is a feature of the **Pulsonix Interactive High Speed option**, it is only available with this license.

Back Drilling, is a process used to remove the unused stub of drill holes from a through-hole when creation high-speed PCBs. This technique ensures signal stubs are minimised (which can be a source of impedance discontinuities and signal reflections). With increased data rates this aids the reduction of signal distortion. This is alternative method to using blind and buried vias which are costly to manufacture.

The process of Back Drilling

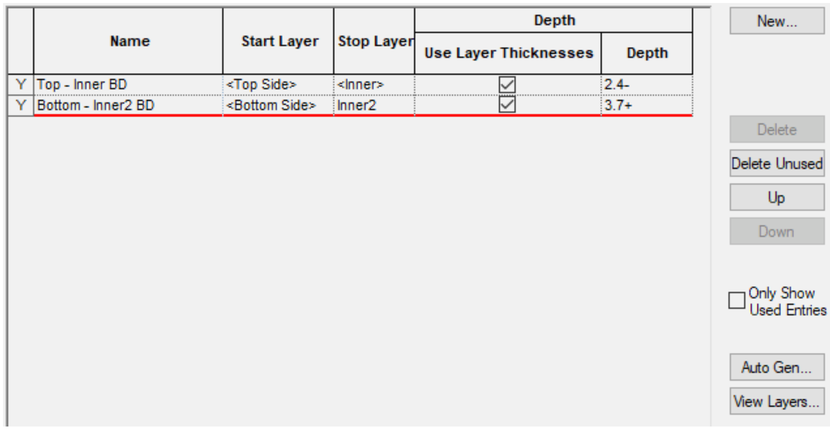
In order to utilise Back Drilling, the summary below informs you of the general process:

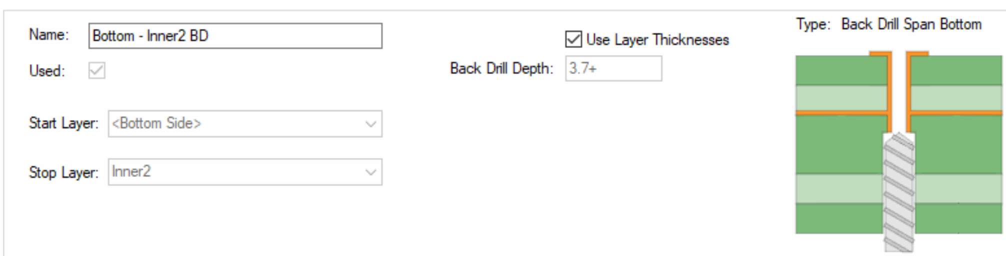
- Identify a requirement for it, then:
- Add Back Drill Spans to accommodate Back Drilling in your Technology
- Create Back Drilling Rules in your Technology
- Route your design, vias and spans that match the Back Drilling rules will be created
- Export the Back Drilled Vias to the Excellon NC Drilling format using the CAM Plot option You can also export a user report created using the Report Maker.

In addition to the above, new features have also been added for Design Rules Checking, exporting to various other formats such as IPC-2581, GenCAD, ODB++ etc. and user defined reports using the new commands within the Report Maker.

Technology Back Drilling Spans

A **Back Drill Span** page has been added to the **Technology** under **Layers**. These spans define the layers of a via that need to be Back Drilled. The start layer of the span is either **Top** side or **Bottom** side, and the end layer will always be an inner layer. All layer spans have to have unique name; a **Layer Span** cannot have the same name as a **Back Drill Span**.





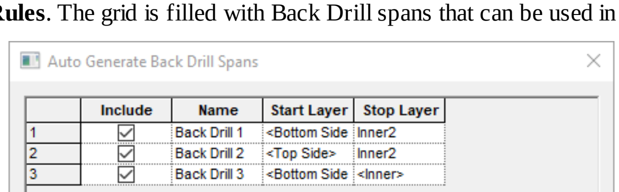
The **Back Drill Depth** is the depth of the oversized drill hole after Back Drilling. This value takes into account the **Back Drill Stub Length** (taken from the **Back Drill Rules** page), and is measured from the top or bottom of the board to the inner electrical 'Stop' layer of the Back Drill span (this does not include the thickness of the stop layer). It is calculated from the layer thicknesses, or can be specified as an explicit value by unchecking this option and typing the value required. If you have specified layers thicknesses, you are likely to want to use these.

If the **Use layer thickness** box is unchecked, you can define your own thickness for the span. When the drill depth is <undefined>, the calculated drill depth is less than zero. The most likely cause of this is when the defined **Back Drill Stub Length** being too large.

The **View Layers** button will display a cross sectional view of the layer stack the same as the **Layers** dialog.

Auto Generating Back Drilling Spans

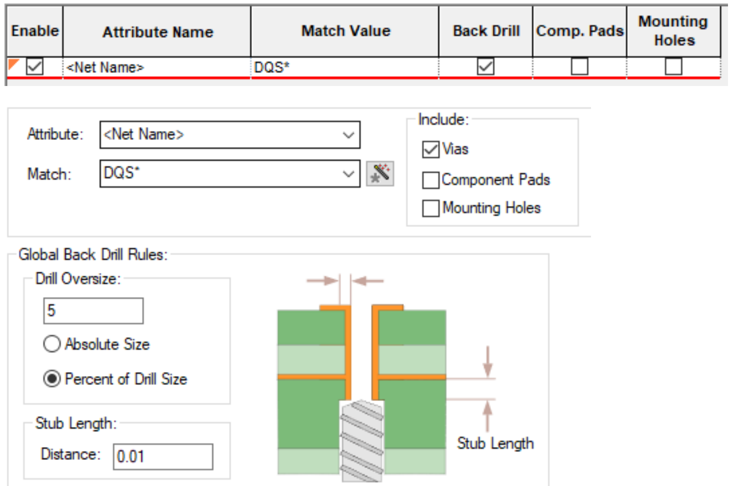
The **Auto Gen** button on this page enables you to automatically generate a list of Back Drill spans that are usable by your current design. These will be generated from the rules defined in the **Back Drill Rules**. The grid is filled with Back Drill spans that can be used in the design.



The check box for whether you **Include** then or not include them enables Back Drill Spans rules used to be overridden if not required.

Technology Back Drilling Rules

The **Back Drill Span** rules page in the **Technology Layers** dialog allows you to define the nets that will be Back Drilled (or not if excluded from the list). A net will match with the first relevant rule in the list, so the order of the rules is important. As with all rules, wildcard matches are permitted.



Use the **Include** check box options to select the **Vias**, **Component Pads** or **Mounting Holes** that will be Back Drilled. The **Vias** check box is always selected but Components and Mounting Holes can be optionally added if required.

The **Global Back Drill Rules** at the bottom of the dialog let you define a **Drill Oversize** for the larger drill used for all Back Drilling in the design.

Drill Oversize will be the increase in the drill radius compared to the vias' defined **Drill Size**, or as a **Percentage** increase in radius compared to the vias' drill size.

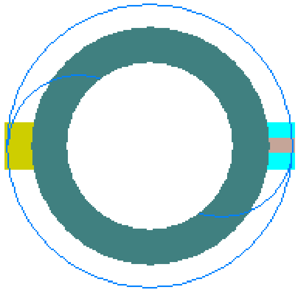
You also define the remaining **Stub Length** required after Back Drilling has been performed. This is your safety margin to absolutely ensure the via stop pad required isn't drilled out.

*The sizes and values for **Drill Oversize** and **Stub Length** will be supplied by your board manufacturer, it is important to define these very carefully to avoid errors.*

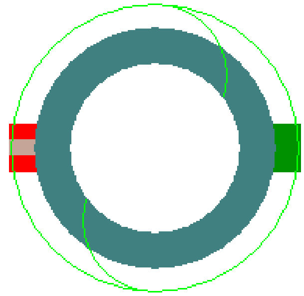
Identifying Back Drilled Vias in your Design

Back Drilled vias have a circle drawn around them (with the same diameter as the oversized drill) and a curved line going through the pad.

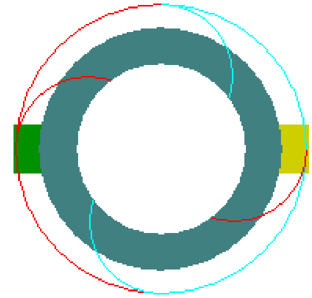
The colour of the circle is dependent on whether it is a Top Back Drill (same colour as Top layer) or a Bottom Back Drill (same colour as Bottom layer), with the curved line being the same colour as the Stop layer of the Back Drill. The curved line will be horizontal for a Top Back Drill and vertical for a Bottom Back Drill. If the via has both a Top and Bottom Back Drill span, then both shapes are drawn together.



Top Back Drill



Bottom Back Drill



Both Top and Bottom Back Drill

Back Drill Colours

The colours for the three Back Drill types can be set using a new tab in PCB **Colours**, called **Back Drill Spans** and by selecting the relevant **Back Drill Span** in the list.

Back Drill	Displayed	Colour
Top > Inner 3	☑	
Top > Inner 4	☑	
Bottom > Inner 6	☑	
Bottom > Inner 7	☑	

Update Back Drills

A new feature named **Update Back Drills** has been added to the **Utilities** menu. Use this option to check and assign the most appropriate Back Drill spans to the vias that are on Back Drilled nets and that currently don’t have this status.

This option is also automatically run when the **CAM Plot** option is initially run.

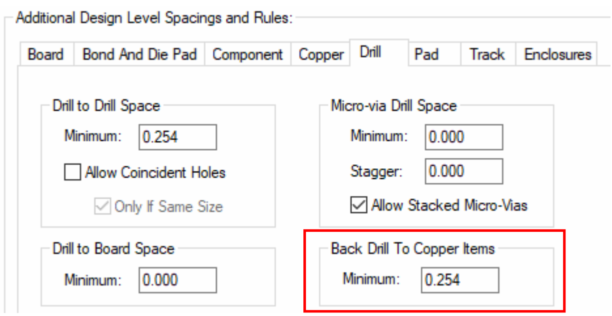
Back Drilled vias can be checked for using new checks in the **DRC** option, see below.

Back Drill Spacings

Back Drills are taken into account when doing both **Design Rules Checking** and **Online DRC**.

A new Drill spacing has been added to the **Design Level Spacing Rules** on the **Drill** tab called **Back Drill to Copper Items**. This is the spacing between Back Drills and other items such as Pads, Vias, Copper etc.

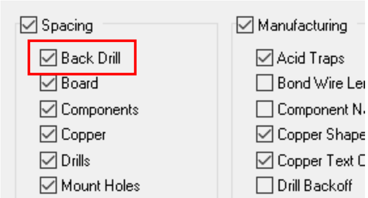
The ‘general’ **Drill to Drill** spacing handles Back Drill to Back Drill spacing, and Drill to Board handles Back Drills to Board spacing.



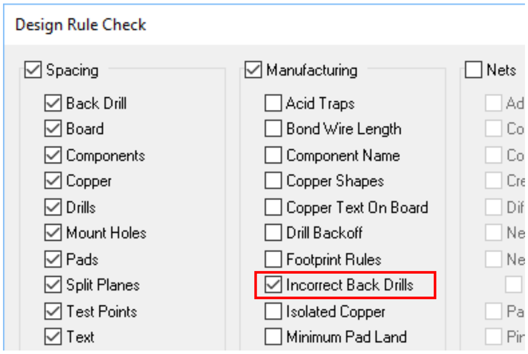
Back Drilling DRC

The **Design Rules Check** dialog now has a **Back Drill** check box in the **Spacing** section.

This checks the spacing between Back Drills and other items using the **Back Drill To Copper Items** spacing defined in your Technology. Back Drill to Board and Back Drill to Drill is checked in the **Drills** Spacing.



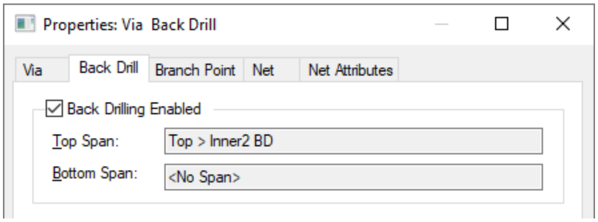
A new check has been added to the **DRC** under the **Manufacturing** section called **Incorrect Back Drills**. This check will report an error if a Via Back Drill span is currently incorrect (checked against the **Back Drill Rules**) and requires updating using the **Update Back Drills** option.



Running this will also check if there are any Back Drill spans that have a negative drill depth. This happens when the **Global Back Drill Stub Length** is greater than the depth of the span itself (as defined in the **Back Drill Rules** page).

Via & Component Pad Properties

The **Via & Component Pad Properties** dialogs now has a **Back Drill** tab which shows the used Back Drill spans. Checking and unchecking the option allows you to enable or disable Back Drilling for that specific via. The Back Drill spans cannot be changed inside Via/Pad Properties.



Back Drilling Pin Depth Attribute for Component Pads

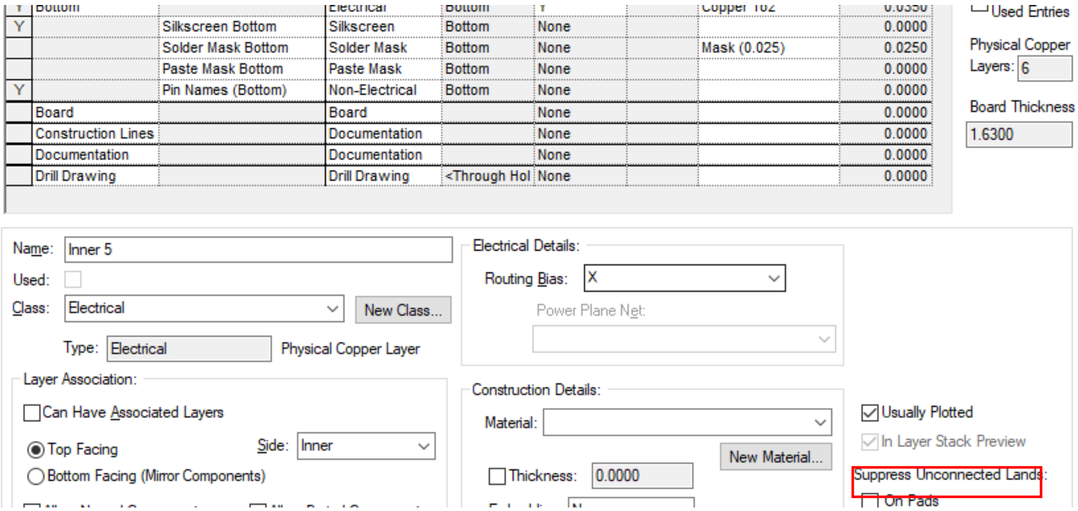
A new attribute on a Component pin called **Pin Depth** has been added. This allows you to define the depth of an individual pin so that you will know the maximum depth allowed for Back Drilling.

A new attribute has been added called **Part Pin Depths** that allows you to define all the pin depths for that specific Component. This means by setting this attribute on a Part, it will not need to be set on every pin of the footprint. This will override any value assigned using the **Pin Depth** attribute.

If a Back Drill rule defined includes Component Pads, neither of these attributes are used.

Layer Span Layers automatically included in Layer Stack Preview

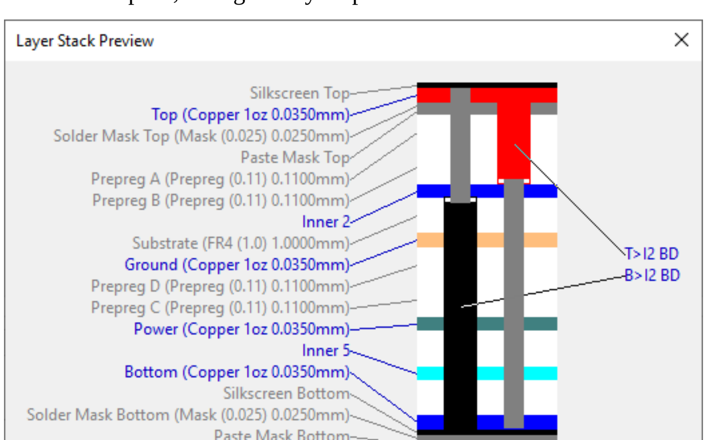
The **From Layer** or **To Layer** of a Layer Span, or the **Stop Layer** of a Back Drill span is now included in the **layer stack preview**. For the applicable layer where these are used, the **In Layer Stack Preview** check box on the **Technology Layers** page will be checked and disabled (greyed out).



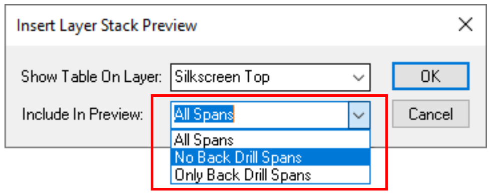
Inserted Layer Stack Previews

Both the **Layer Stack Preview** inserted into the design and the **View Layers** option (in **Technology, Layers**, **Layer Spans** and **Back Drill Spans**) now show Back Drills.

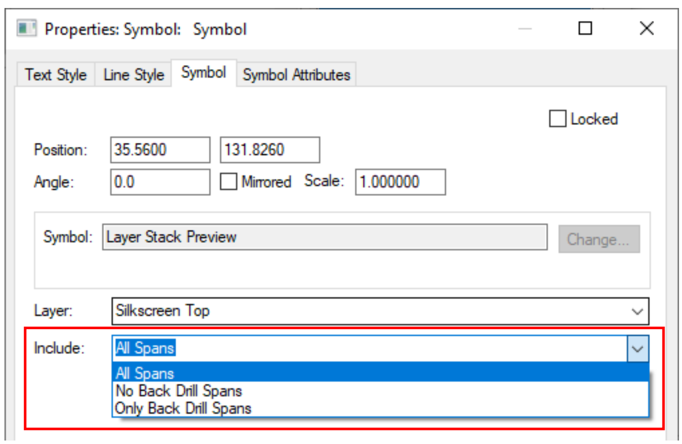
In the Technology, the **View Layers** button works the same on the Layers and Layer Spans pages (shows layers and regular layer spans) but on the Back Drill span page, it will show the layers along with the Back Drill spans, no regular layer spans are shown.



On the **Insert Layer Stack Preview** dialog, there is now an **Include In Preview** option. This defines which spans are included in the preview – **All spans, No Back Drill spans,** or **Only Back Drill spans**.



This can also be changed later by changing the newly added include option in **Symbol Properties** for the selected **layer stack preview**.



When changing the text box fill colour of a **Callout** on a **Layer Stack Preview**, the outline of the span will also be in the selected colour – this was changed to compliment the Back Drill span shape in layer stack preview.

CAM Plot Wizard

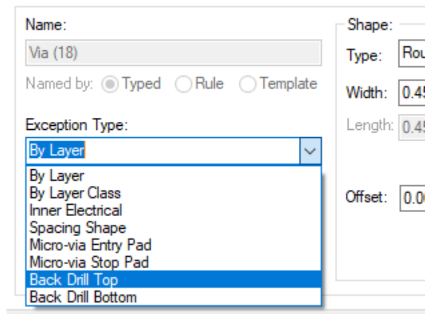
In the **CAM Plot Wizard**, you can now select **Back Drill spans** as a process, allowing for the creation of drill files and drill drawings.

The insert drill table dialog now allows for Back Drill span drill tables. For values to be correct the Back Drill span must be used by a via, and a drill oversize must be defined.

Pad Exceptions for Back Drilling

When creating a pad exception, Back Drill Top and Back Drill Bottom **Exception Types** can be used. This allows for a pad exception to be added on a via that will be Back Drilled, on layer top or bottom depending on the exception type.

The Exception Type allows you to select **Back Drill Top** or **Bottom** from the list:



Excellon Report Chanegs

The Back Drill **depth** and Back Drill **side** are both output in the Excellon report. The drill output will always be unplated.

```
Excellon Drill Output
Output File: C:\Designs\ExcellonPlots\Job66(Excellon).drl

    Plot Name:        Excellon
    Variant:          <Master Design>
    Contains:         Bottom >Inner 2 BD           Unplated
    Side:             Bottom
    Drill Depth:      0.0500

    Drill Type:       Unplated
    Scale:            1.000
    Mirrored:         No
    Rotated:          No
    Offset:           -41.9040    -65.1905mm
    Step & Repeat:    Not used
```

IPC-2581 and ODB++ Exports include Back Drills

Back Drills are now included in IPC-2581 and ODB++ exports. No extra selection is required, this is automatically done for you.

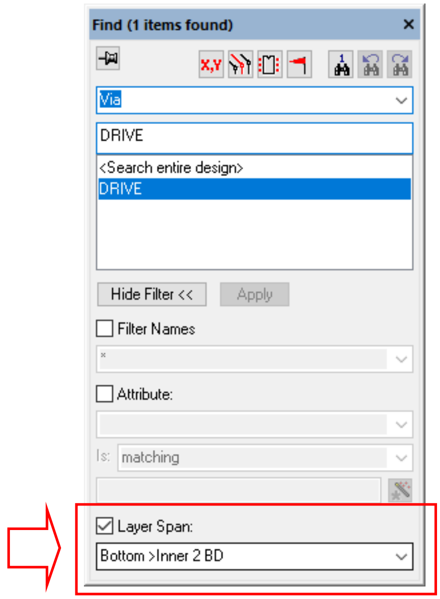
Back Drilling Commands in the Report Maker

New commands have been added so that Back Drilled Vias can be reported and/or added as a user report to the design to display manufacturing information about them.

For more detailed information, see section below *Report Maker Changes*.

Find Bar

You can use the **Find Bar** to locate any vias that currently have a Back Drill span. When searching for vias, check the **Layer Span** box at the bottom of the bar, and select a Back Drill span from the drop down list.

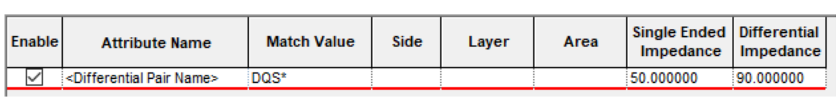


Track Impedance Rules

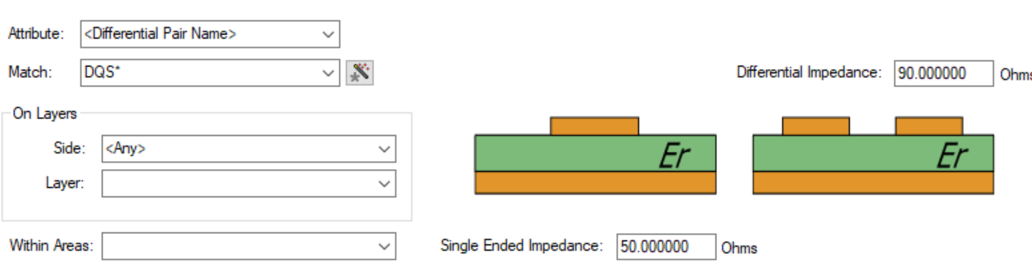
A new rule in the **Technology** under **High Speed** rules, **Track Impedance**, enables you to assign **Single-Ended Track Impedance** and/or **Differential Track Impedance** (also called Edge-Coupled Impedance) to nets or Differential Pairs on defined layers, and optionally within named areas. The **Design Calculator** will search the rules for the impedance for a selected track.

These rules will be set up when **Importing an IPC2581 Layer Stack** and used by the **Design Calculators** and **Track Impedance Calculator** (when calculating Track Width).

When using this dialog, the **Attribute** selection is made using the drop-down list where a net or attribute can be defined, this also include <Differential Pair Names>.

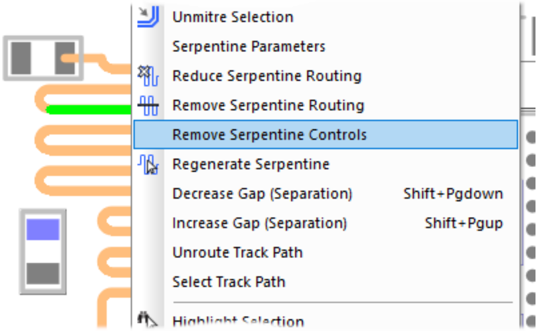


Enable	Attribute Name	Match Value	Side	Layer	Area	Single Ended Impedance	Differential Impedance
☑	<Differential Pair Name>	DQS*				50.000000	90.000000



Remove Serpentine Controls Option

You can now remove the controls on a selected Serpentine track using the **Remove Serpentine Controls** option from the context menu. This reverts it to a normal track that can be edited in the usual way. Once this is done, it cannot be converted back to a controlled Serpentine (although Reduce and Remove Serpentine routing commands may still work as Pulsonix recognises serpentine patterns).



Rules Spreadsheet Multi-threading

The **Rules Spreadsheet** can now be filled using multiple threads when not using the **Only show selected** or **Dynamic item** options. This will speed up filling for designs with large nets or where the design contains many nets.

The selection to **Enable Threads** for the **Rules Spreadsheet** has been added to the **Options** dialog, **General** page. This setting is used by all Multi-threading technology in Pulsonix.

Create Differential Pairs From Rules

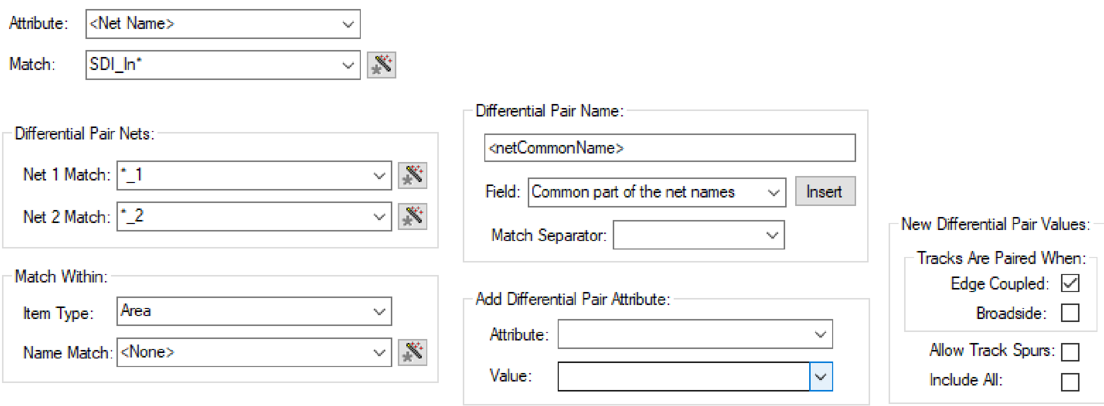
You can now create Differential Pairs in a Schematic or PCB design by setting up and then applying new **Differential Pair Nets** rules, defined in the **High Speed Rules** section of the **Technology** dialog. This enables you to create a rule or rules that capture the net names in your design to apply to nets that should become **Differential Pairs** without having to do this process manually.

Each rule defines a pair of net match strings that are used to find two nets that match the strings where the common part of the name is the same.

The **Attribute** value, Net Name for example, will be the **Match** criteria. You can set this to a specific match, such as SDQ0* or can be more generic, such as * for the whole name.

Once the attribute Match has been satisfied, the rule then uses the Differential Pair Nets rule to make the pairing. For example, rule ***_P** and ***_N** will match nets **SDQ0_P** and **SDQ0_N**. When two nets are matched, the rule defines what name the Differential Pair will take, whether an attribute should be added to use when allocating length rules, and some default parameters for the pair.

Enable	Attribute Name	Match Value	Differential Pair Nets		Match Within		Template String	Edge Coupled	Broadside	Allow Spurs	Include All	Add Differential Pair Attribute	
			Net 1 Match	Net 2 Match	Item Type	Name Match						Attribute Name	Attribute Value
☑	<Net Name>	DQS_*	*P	*N	Area	<None>	<netCommonName>	☑	☐	☐	☐		
☑	<Net Name>	LCO_D*	*1	*2	Area	<None>	<netCommonName>	☑	☐	☐	☐		
☑	<Net Name>	SDI_In*	*_1	*_2	Area	<None>	<netCommonName>	☑	☐	☐	☐		
☑	<Net Name>	SDQ0*	*_P	*_N	Area	<None>	<netCommonName>	☑	☐	☐	☐		



Attribute

A rule defines two sets of nets. Each set is defined using a net **Attribute Name** and a **Match** value. The Match Value can be wildcard. In particular, you can match the inbuilt attributes **Net Class Name** or **Net Name**. When these checks are done, each rule is applied, so the order of these rules is important.

Match

The Match value, Net Name for example, will be the Match criteria use for the rule. You can set this to a specific match, such as SDQ0* or can be more generic, such as *. Only nets matching this value will be considered for a Differential Pair.

Differential Pair Nets

Once the **Attribute Match** has been satisfied, the rule then uses the Differential Pair Nets rule to make the pairing. For example, rule *_P and *_N will match nets SDQ0_P and SDQ0_N. When two nets are matched, the rule defines what name the Differential Pair will take, whether an attribute should be added to use when allocating length rules, and additional default parameters for the pair.

The Net 1 Match

This is used to match the net defining one 'half' of the Differential Pair net. It is matched along with the Attribute Name. The Match Value can be a wildcarded string. For example, if the Attribute name matches SDQ0, the net 1 match might be *_P so will match SDQ0_P.

The Net 2 Match

This is used to match the net defining the second 'half' of the Differential Pair net. It is also matched along with the Attribute Name. The Match Value can be a wildcarded string. For example, if the Attribute name matches nets SDQ0*, the net 2 match might be *_N so will match SDQ0_N.

Match Within (for a PCB design)

You can also choose to only match nets if they have pads within a certain area. The **Item Type** will show **Area** or <None> when used within a PCB design. The **Name Match** can be an area name selected from the drop-down list or it can be left blank so the rule is applied to the whole design. The Match Value can be wildcarded.

Differential Pair Name

This box shows you how the Differential Pair Name will be constructed if you let the system create one for you. This will be based on the Field and Match Separator fields below. This is an editable box and can be constructed from the Fields selected form the drop-down list. At any point, you can also use the **Delete** key on your keyboard to removed fields if not required.

The **Field** drop-down has a list of possible contents from which to create the Differential Pair Name. Possible contents available are:

Common part of the net names - use this to name the Differential Pair with the common part of the two net names. If the Attribute Match string contains a stem before the wildcard * you can choose a separator to be used after that stem in the resultant name. If this is required, type the **Match Separator** or select it from the drop-down list. For example, if the Attribute Match string is SDQ* and *_P and *_N are used to match nets SDQ0_P and SDQ0_N, then the common part of the names will be SDQ0, and if the Match Separator is "_" it inserted after the stem to give a Differential Pair name of SDQ_0.

First net name

Second Net Name

Net Attribute - select the net attribute name from the list presented. The value will be extracted from the first net used to define the Differential Pair.

Component Attribute - select the component attribute name from the list presented. The value will be extracted from the first of the pins used to define the Differential Pair.

The **Insert** button takes the Field (and Match Separator if selected) and passes them into the Differential Pair Name to show you how it will be constructed in the rule when used. Multiple fields can be selected from the list and inserted. The insertion will depend on your cursor position in the Name field above.

Add Differential Pair Attribute

Use this to define an attribute to be added to the newly created Differential Pairs using this rule. For example, to assign an attribute to be used to allocate a Differential Pair Gap or Skew rule. Leave the fields blank if no attribute is to be added.

If required, select the **General Use** or **Net Attribute Name** from the drop-down list, and type the Value to be added. If some design items already have attributes using this attribute name their values can be selected from the Value drop-down list.

New Differential Pair Values

You can define how the tracks are paired. **Edge Coupled** are the usual Differential Pairs, the edges of the two tracks separated by the specified gap. **Broadside** are paired vertically, the tracks laid on top of each other on different layers. It is possible to allow both types of pairing on a Differential Pair.

Allow Track Spurs

Normally, the path between the pins in a Differential Pair should be without any spurs or branches to other pads, vias, etc. Checking the **Allow Track Spurs** option will allow spurs from the track path. These spurs must not have any further spurs or branches and should be terminated on a pad, via or testpoint. These will be checked as part of the **Differential Pair Design Rule Check** and in the Differential Pair report.

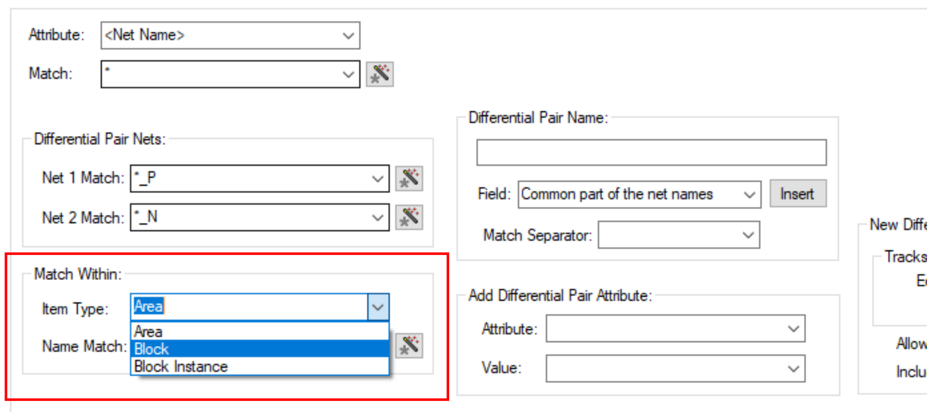
Include All

Normally pins are not included in the path between the pins at the ends of the Differential Pair, but sometimes this is required. Checking the **Include All** box will instruct **Optimise Nets** to connect to any extra pins on the net that are in this path and not assigned to other Differential Pairs, allowing you to route them into the path without a Differential Pair error.

Match Within Schematic

If setting the **Match Within** value from a Schematic, the drop-down list will also display additional settings for **Block** and **Block Instance**.

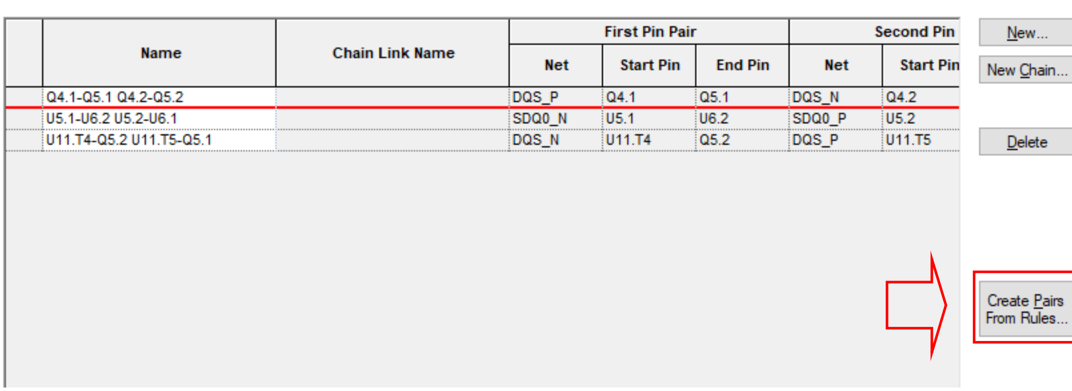
In this case, Block Instance Name is the name of an instance of a block in the design. Block Name is the name of a block used by the instance; several block instances may use the same named block. The nets may be across the design but the reason for the area is only pins defined in that area, block, or block instance will be considered when creating the Differential Pairs.



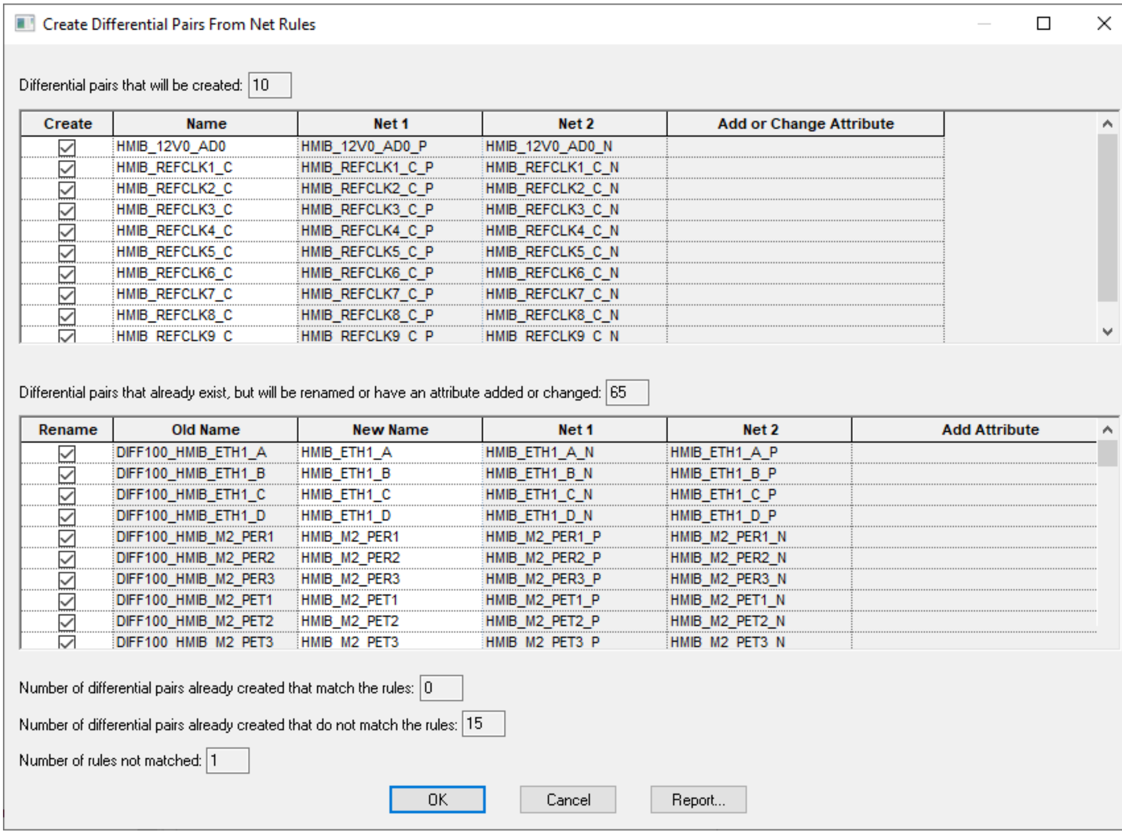
Create Pairs From Rules Dialog

When the rules have been set up, use the new **Create Pairs From Rules** button on the **Technology**, **Differential Pairs** page. You will be presented with a dialog that informs you which new pairs will be added and which existing pairs will be renamed or have an attribute added due to the rules. Here you can choose to not add or rename a Differential Pair by unchecking it in the list, or can type their own name for the pair.

The New Chain dialog will not create a Differential Pair if the nets do not have enough nodes.



In the Technology dialog, if you select an existing Differential Pair Name or add a new one and it matches a rule that would give a different name or would add an attribute, a button will appear next to the **Name** field to use to apply the rule. This will change its name and/or add an attribute.



The dialog is split into two sections; Differential Pairs that will be created and Differential Pairs that already exist, but will be renamed or have an attribute added or changed.

Differential Pairs that will be created

The number next to this header informs you of the number of potential new Differential Pairs that will be added to your design if all the rules available are applied. The main list is a summary of all the Differential Pair names and their paired nets. By default, the **Create** check box next to each name will be selected (so that all new Differential Pair Names required will be added). If the new name is blank then the name will be the default name generated from the four pins names. Uncheck this box if you do not wish to create a new pair in your design. If you do this, care should be taken to ensure you haven't missed a pair that you defined in the rules.

This list is applied when the **OK** button is pressed at the bottom of the dialog.

Differential Pair that already exist

This reports any matches between the rules defined and the Differential Paired nets already in the design. The number displayed next to the header is a match based on the rules defined. Using the Rename check boxes, you can select which existing differential Pairs to rename. Again, care should be observed if you uncheck any of these as it means they match rules defined but you are choosing to ignore them.

This list is applied when the **OK** button is pressed at the bottom of the dialog.

Summary boxes

Three summary boxes at the bottom of the dialog report an overall position to you based on the check boxes selected in the grids above, any rules that it fails to match based on the Differential Pair Rules defined and a count of the number of rules that were not matched.

Number of Differential Pairs already created that match the rules, shows the number of Differential Pairs which match your defined rules and that exist in your design. This is shown as a reference for you.

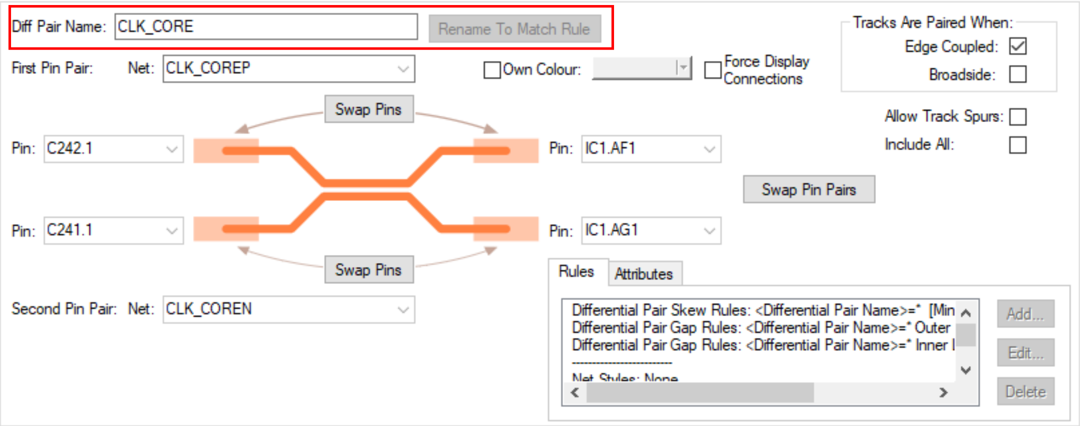
Number of Differential Pairs already created that do not match the rules, shows how many Differential Pairs that are currently in your design that do not match any rules. This is a very useful check to ensure that if you do want to cover all Differential Pairs using rules, then this should read zero. If the number 1 or more is shown then there are Pairs that are not covered using a rule.

The **Number of rules that were not matched** shows that Differential Pairs rules have been created but there are no matches in the design to satisfy this.

Use the **Report** button to report listing the Differential Pairs that already matched, did not match at all, and the rules that were not matched (with an explanation of why).

Reapplying Rules

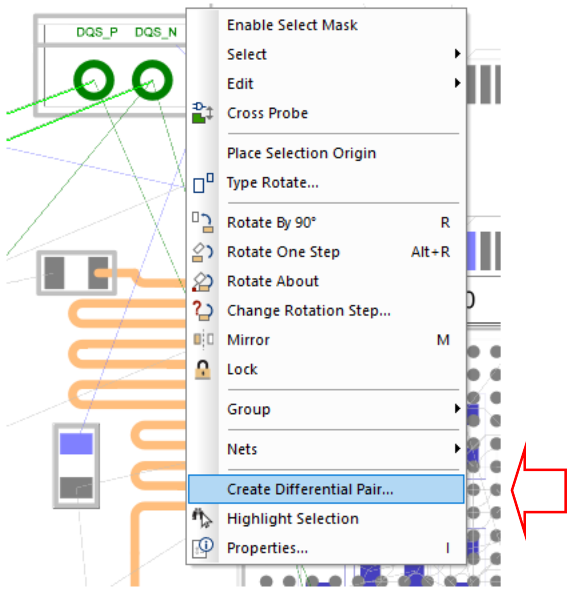
A new button on the **Differential Pairs Nets** dialog next to the **Diff Pair Name** enables you to **Rename to Apply Rule** to match any new rules defined in the **Differential Pair Net Rules** dialog.



Create Differential Pair using Context Menu

You can now select two nets in a PCB or Schematic design and from the context menu use **Create Differential Pair** to create the 'pair'. Once these nets are selected, it will open up the **Differential Pair Rules** dialog in your **Technology** with these nets pre-selected and ready to define.

If four pads are selected, they will be used for the pair, otherwise the pads will be automatically allocated. You will not see this command on the context menu if the nets already form a Differential Pair, or don't have at least two pins on each net.

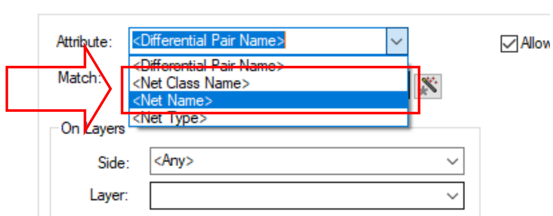


Define Differential Pair Rules using Net and Net Class Attribute

You can now define **Differential Pair Gap** and **Skew** rules with a **Net or Net Class attribute**. The rule is only used on a Differential Pairs where both (paired) nets match the attribute. For example, defined for a specific Net Class name.

Previously, it was only the Differential Pair name or Differential Pair attribute that could be used, now you can assign a Net attribute and it will check both nets for a match.

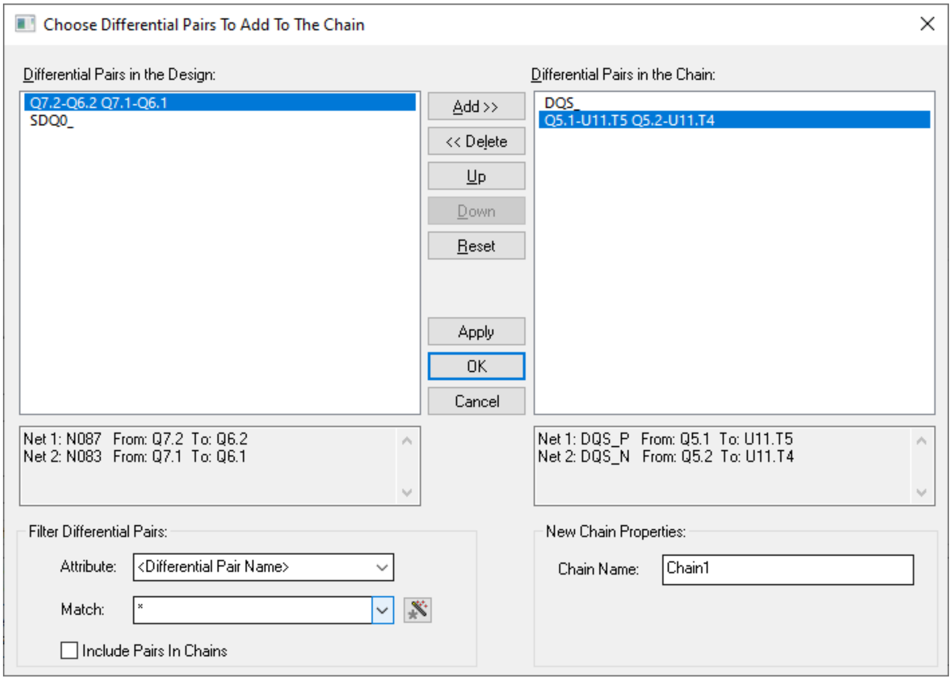
Note: Differential Pair Skew rules defined this way will not be used on Chains (which can have multiple nets), only on Differential Pairs.



This feature was back-fitted to 10.5

Create Differential Pair Chain

The **New Chain** dialog on the **Differential Pairs** dialog has changed to make it easier to choose the correct Differential Pairs for a chain.



It now features two lists; the left one contains a filtered list of Differential Pairs in the design to choose from, and the right list contains the chosen Differential Pairs that will represent the chain.

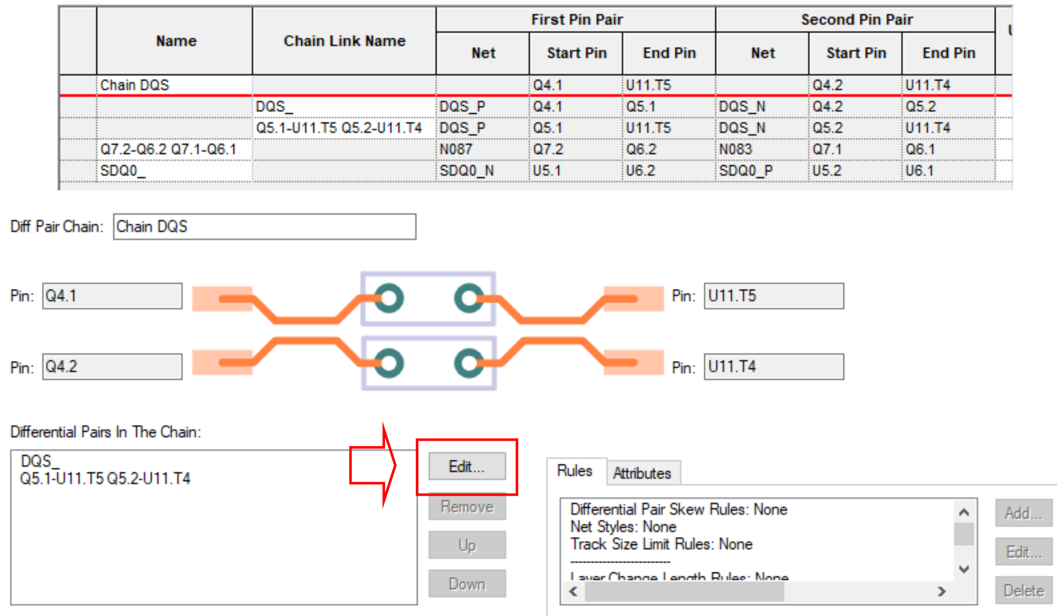
Selecting a Differential Pair name in ether list will display its nets and pins in an information panel below the list. You can filter the left list by attribute that can be a **Differential Pair attribute**, **Net attribute**, **Component attribute** or **Pin attribute**. The Net attribute will have to exist on either Net on the pair and the Component or Pin attribute will have to exist on any of the four pins in the pair.

You can also **Include Pairs In Chains** (or exclude by unchecking the option) all Differential Pairs that are already in a chain from the list, this helps refine the list.

The **Chain Name** text box allows you to set a name for the newly created Differential Pair chain when you exit this dialog. If this box is left blank, then a default unique chain name will be generated for you.

The **Apply** button allows you to quickly create a chain and start the creation of another chain. This is done by creating a chain using the currently input details. Checking on Apply will ensure if name is unique, if not enough Differential Pair have been added, etc. It then then resets the dialog and updates source list to allow you to start the creation of another Differential Pair chain without you needing to reopen the dialog using the **New Chain** button again.

Once you have the Differential Pairs selected, type a new Chain Name and press **OK** to return to the Differential Pairs dialog where the chain will now be presented:



Once you have created the chain, selecting it in the grid still displays the chain controls, but the **Add…** button has been replaced with an **Edit…** button. Selecting it displays the New Chain dialog where you can edit the chain.

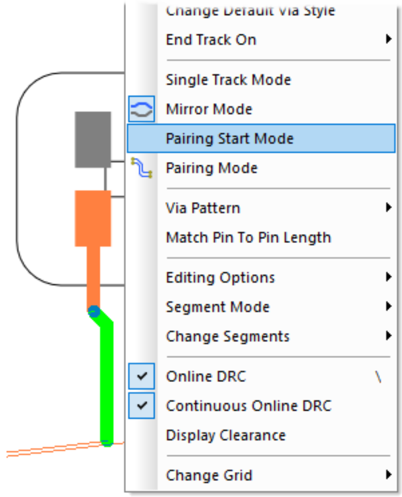
New <Component Pin Name> attribute

To make this chain easier to locate, a new pin attribute has been created **<Component Pin Name>,** for example **U12.6**. Use this if you need to find the pair that connects to that pin. This attribute is available throughout Pulsonix.

New 'Start Pairing' Mode for Differential Pairs

The ability to start routing a Differential Pairs using new **Pairing Start Mode** instead of using **Mirror Mode** has been added.

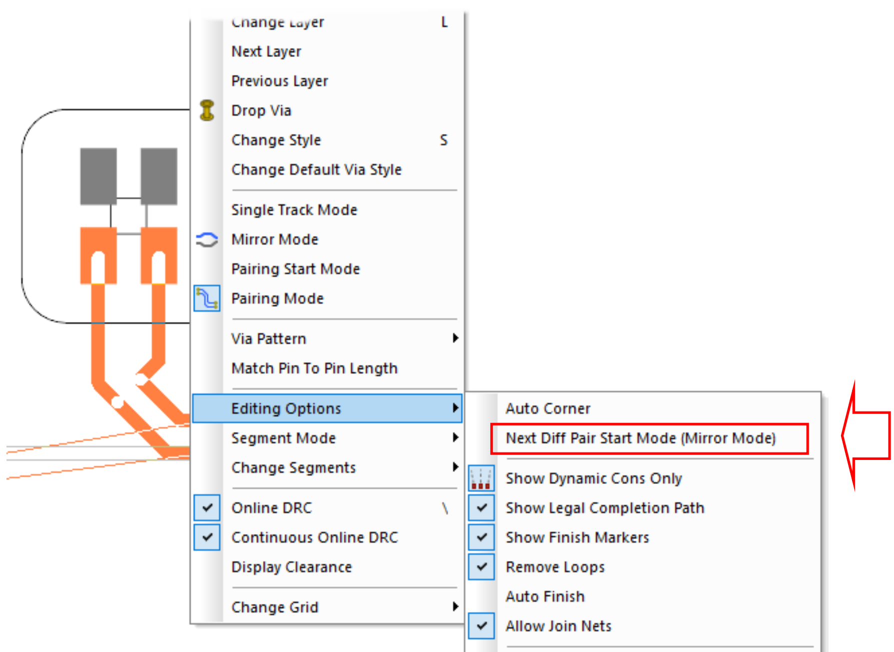
When you start editing a Differential Pair track, the mode used will be as defined in the **Options Edit Track** page.



The choice is **Single Track Mode**, **Mirror Mode** and the new **Pairing Start Mode**.

Paring Start mode is fundamentally the same as using the **Insert Bus Route** option in that it gathers the two tracks by single side routing to the two junctions representing the start of the paired section. Moving the cursor moves the junction pair to the required position, and left click starts the pairing. You can switch between the three modes using the context menu, which will show which mode you are in.

Whilst editing, you can check what auto start mode is currently set using the **Editing Options** sub menu from the context menu. The mode is shown in brackets.

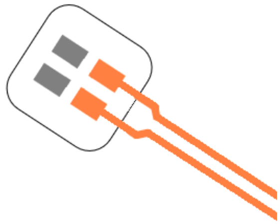


Choosing **Next Diff Pair Start Mode** cycles to the next mode from the one shown. This replaces the **Start Mirroring Paired Track** option in 10.5.

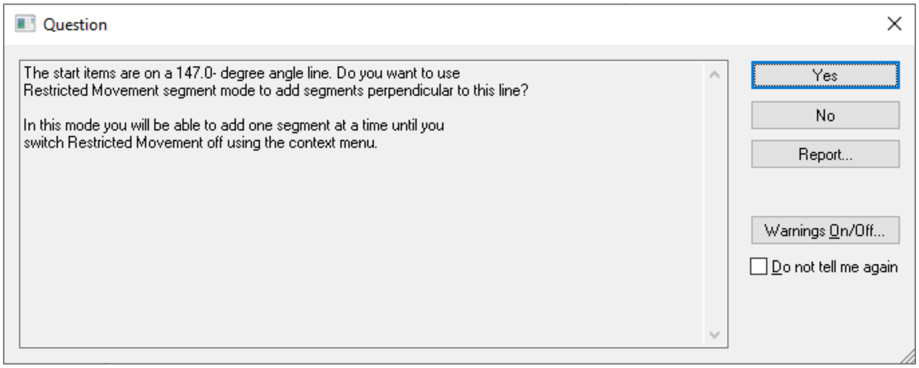
Whilst adding paired segments, you can end pairing by selecting one of the three modes to carry on editing. Using **Pairing Start mode** here allows you to automatically route the tracks around an obstacle and start pairing again.

Differential Pair Mirror Mode

When using **Mirror Mode** and mirroring paired tracks (Differential Pairs), the initial segments and the mirror line is now forced to be perpendicular to the line through the pads. Previously, the angle of this line was based on where you clicked to start the track. This mainly effects Differential Pairs that start on angled pads.



If you start on an angled pad, one that isn't 45-degrees, a message is displayed informing you of this:



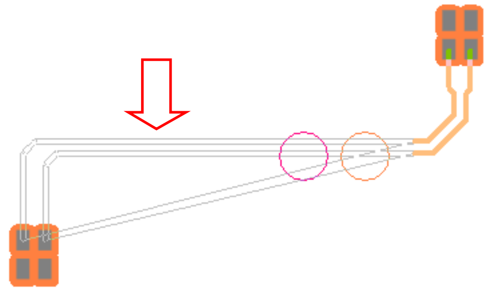
If you want to change the angle of the mirror line use the new **Rotate Mirror Angle** option from the context menu if it is available.

If you did not intend to start mirroring this time and only want to edit the single track, switch to **Single Track Mode** from the context menu.

Legal Completion Path for Differential Pairs

The **Show Legal Completion Paths** option now works when pairing tracks for a Differential Pair. The path shown is as if using pairing start mode from the target pads to the junctions at the end of the moving paired tracks. Using **Complete As Track** will convert these paths to tracks.

This mode also now works when in **Mirror** mode.

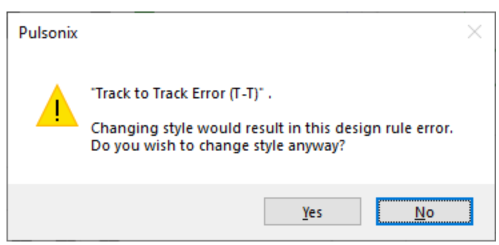


Change Style of Differential Pairs

You can now change the style of paired Differential Pair tracks if multiple tracks are selected. The style of both paired Tracks will be changed, along the entire paired section.

Do this by changing the style in the **Change Style** dialog, or in **Track Properties** (the style combo box is no longer disabled).

If you select a track style that would create a DRC error, a warning dialog is displayed:



Shift to add corner in Differential Pair Mirror Mode

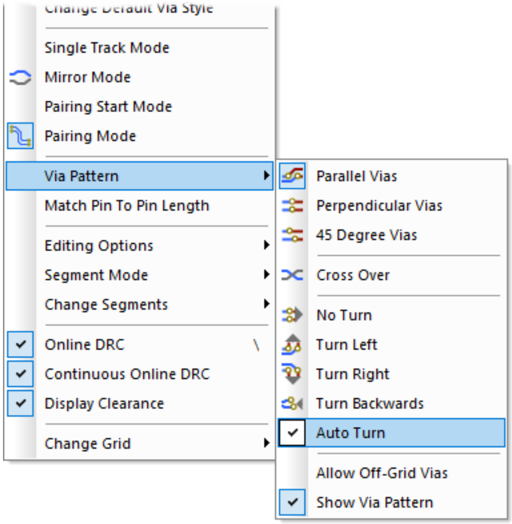
When using **Mirror Mode** and mirroring paired tracks (Differential Pairs), adding a corner when the tracks are close together ends the mirroring mode and starts editing the pair. If you now hold the Shift when adding the corner, you will remain in Mirror Mode. This enables you to add multiple corners to design the track shape required before the pairing starts.

This feature was back-fitted to 10.5

New Via Pattern option - Auto Turn

There is a new **Via Pattern** option available - **Auto Turn**. When changing layers, use this to initially exit the vias straight, but as you move the cursor after vias are added, the pattern will change to exit left, right or backwards to best match the cursor position. Once you corner or backspace, the Auto Turn mode will be switched off. This is now the default action.

Select **Via Pattern>** and **Auto Turn** from the context menu:

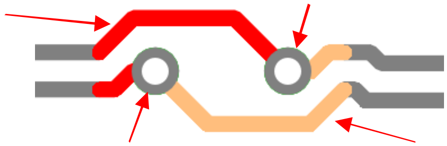


Use copy of existing Differential Pair Via Pattern

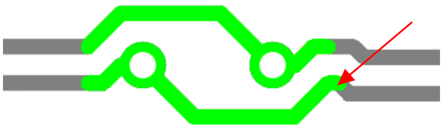
You can now choose to use a copy of an existing Differential Pair Via pattern when changing layer whilst adding paired tracks. You may have already created a pattern that you want to reuse, this can be copied and reused.

Before you start Differential Pair routing you have to choose the pattern to use. To do this first select all items in an existing pattern from the end of the paired tracks on one layer to the start of the paired tracks on the other layer. Using **Frame Select** with the **Select If completely Framed** option enabled will help ensure precise selection. Then, if needed, add any Ground vias that have been placed around the pattern (used for shielding) to this selection.

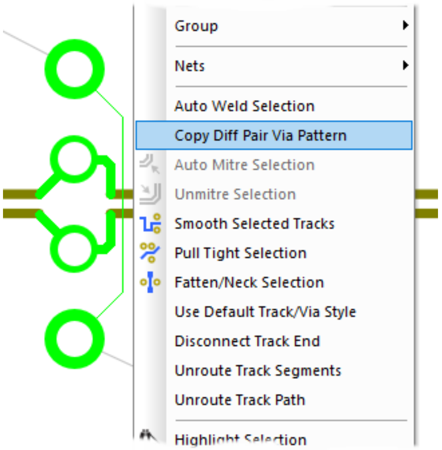
In the example below, selecting the red and yellow tracks plus the vias would be suitable:



In the example below, the selection is more extended to include one segment of the Differential Pair, this is no longer eligible for the Copy option to appear on the context menu.

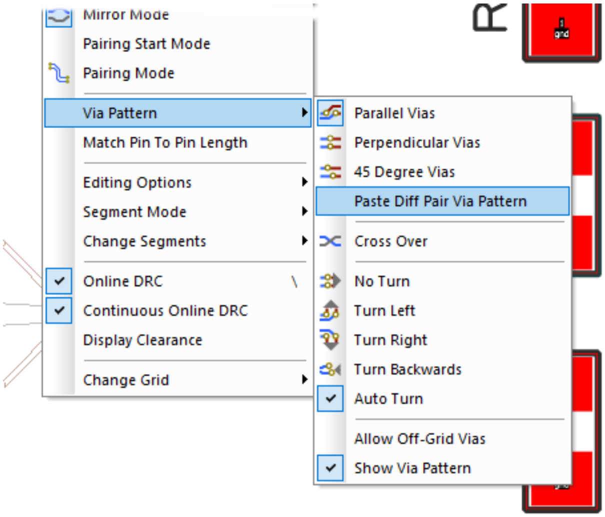


If the pattern is as required, you can then use **Copy Diff completion Pair Pattern** from the context menu to save it to a special location on the clipboard.



The pattern must contain Tracks and Vias on two nets, each net selection must contain a track that starts from a paired track on one layer, and then a continuous path from it to a track that ends on a paired track on another layer. This path can contain several micro-vias. The pattern can change direction and cross over.

Then, when adding new paired tracks, you can now right click to use the **Via Pattern** sub-menu and choose **Paste Diff Pair Via Pattern**.



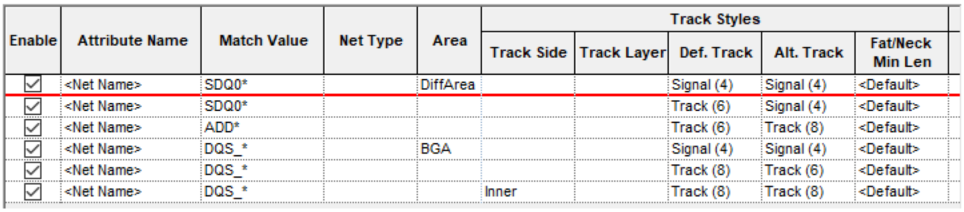
If the copied via pattern is suitable for this pair you will see all its vias as you move the paired tracks. Change layer and the copied via pattern will be inserted in order for you to change layer. If the pattern is not suitable (wrong gap between tracks, wrong layer, etc) then change layer will not be available until you switch the via pattern.

The via pattern option **Paste Diff Pair Via Pattern** will be remembered and will be used in future whilst the pattern is available on the clipboard. Changing layer back to the start layer will use the same pattern rotated about 180 degrees.

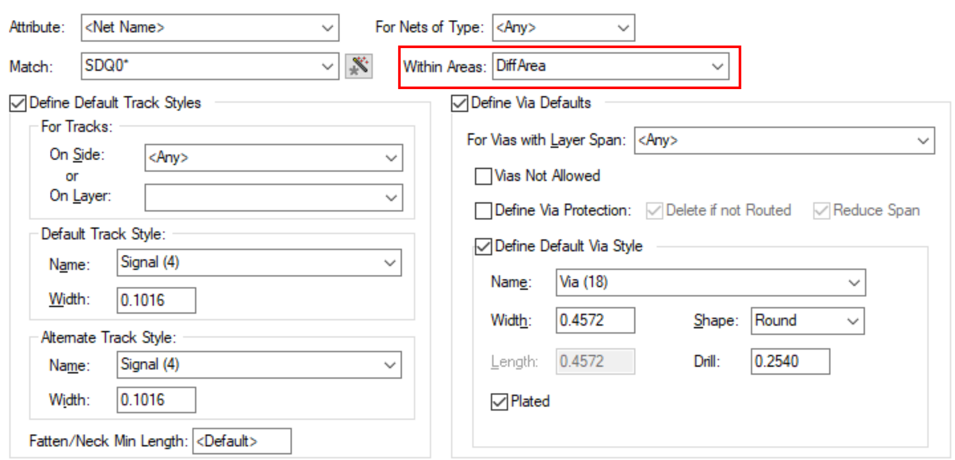
You cannot use **Auto Turn** or **Drop Via** with this feature. When you use change layer it is fixed to the layer that the chosen pattern ends on.

Editing Differential Pairs into Areas

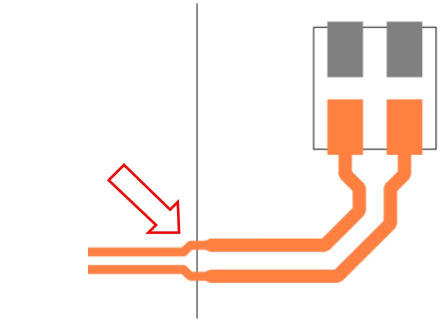
When adding Differential Paired tracks and you cross an **Area** boundary that has different Track styles or Differential Pair gaps defined (in **Net Styles**), the pairing is automatically stopped on the near side and started again on the far side with two transition tracks added over the border joining the ends of the paired sections together. If it was a mistake entering the area, use the **Backspace** key to remove the area transition Tracks and revert to editing the original paired tracks.



Enable	Attribute Name	Match Value	Net Type	Area	Track Styles				
					Track Side	Track Layer	Def. Track	Alt. Track	Fat/Neck Min Len
☑	<Net Name>	SDQ0*		DiffArea			Signal (4)	Signal (4)	<Default>
☑	<Net Name>	SDQ0*					Track (6)	Signal (4)	<Default>
☑	<Net Name>	ADD*					Track (6)	Track (8)	<Default>
☑	<Net Name>	DQS_*		BGA			Signal (4)	Signal (4)	<Default>
☑	<Net Name>	DQS_*					Track (8)	Track (6)	<Default>
☑	<Net Name>	DQS_*			Inner		Track (8)	Track (8)	<Default>



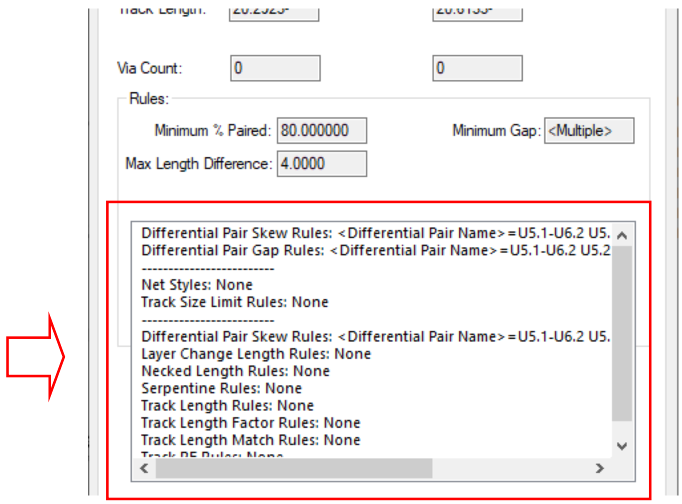
As the Differential Pair enters the named **Area**, it changes to the styles defined in the **Net Styles** dialog.



This feature was back-fitted to 10.5

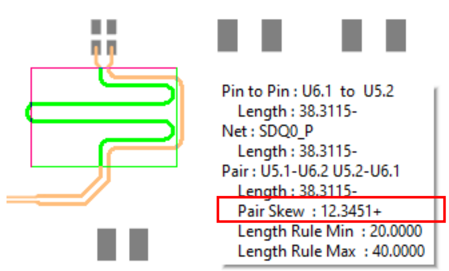
Differential Pair Properties

For completeness the **Differential Pair Properties** tab when selecting a Differential Pair track, now shows a list of Differential Pair Rules (like the **Differential Pair Technology** page).



Differential Pair Track Limits Display

The display for Differential Pair editing now always shows the **Skew Min** and **Max** limits as well as the other rule min and max limits. Previously, it only showed skew limits if it was the min or max limit, and if it was it did not show net length limits (for example).



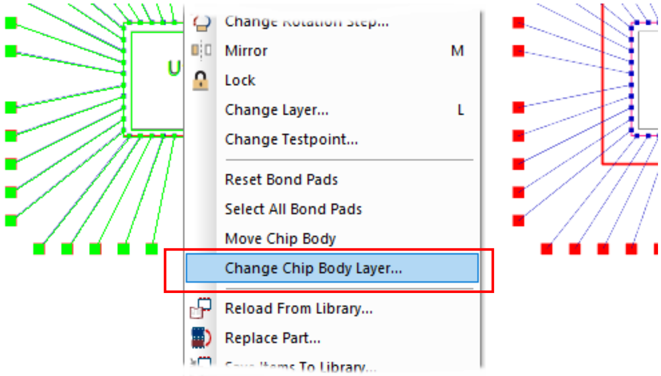
This feature was back-fitted to 10.5

Chip-On-Board Option Changes

The Chip-On-Board options below are only available if you have the Chip Toolkit license.

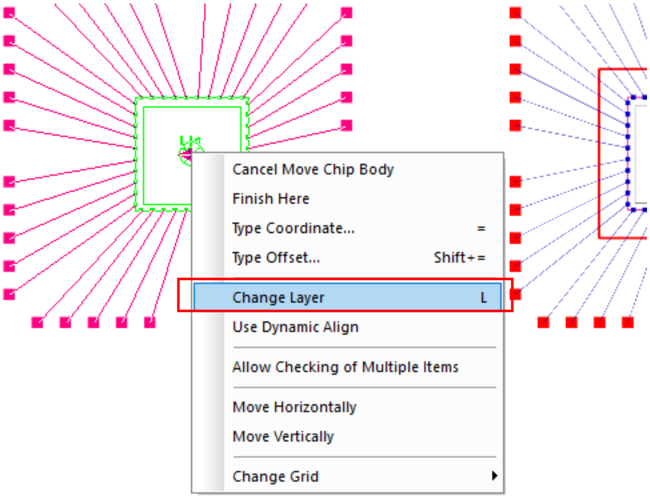
Change Chip Body Layer

With a Chip-On-Board component with bond pads selected, if you right click, there is a new option, **Change Chip Body Layer** available on the context menu. Use this to move the body of the chip down into a cavity, whilst leaving the bond pads on the top side. This avoids the need to move the chip body and then change layer whilst moving it. This option is only available if the design has a suitable layer to move the component to that allows normal components on it.



Change Layer of Chip Body during Move

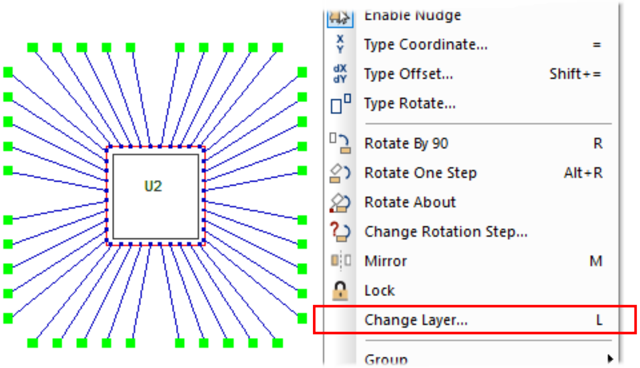
When moving a Chip-On-Board component, using the context menu option **Move Chip Body**, you can now use **Change Layer** from the context to move the component body into a cavity. The Bond Pads layer is not changed.



This feature was back-fitted to 10.5

Change Layer of Chip Bond Pads

In a PCB design for a **Chip-On-Board** component, with **All Bond Pads** selected (using the interactive mode, **Select All Bond Pads** from the context menu) and interactively editable, you can now use **Change Layer** from the context menu to move them to any electrical layer including layers that are not the same as the Die/Die Pad layer.

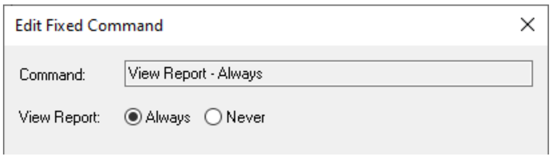


This feature was back-fitted to 10.5

Report Maker Changes

New top-level commands for Append Report & View Report

Two new commands have been added to the top level of the Report Maker - **Append Report** and **View Report**. Both commands can take the value **Always** or **Never**. They override the **Reports** dialog settings for these values and enable you to suppress a report if you wish to run it silent or if the report visibility isn't required (running a format file from CAM Plots for example or within a Script). If using the **Test** button, this command is ignored. It is available when running live reports.



New commands to support Back Drilling

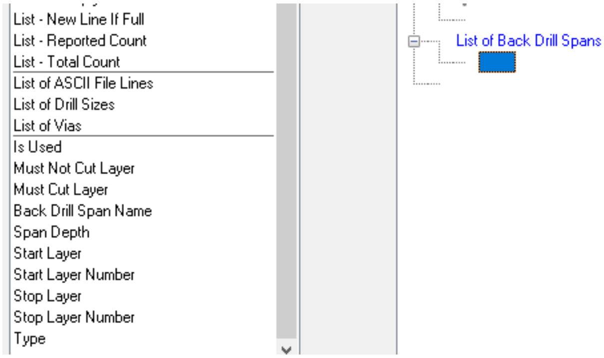
Additional commands have been added to **List of Vias** and **List of Pins** to support the new **Back Drilling** feature:

Is Back Drilled

Top Back Drill Span Name

Bottom Back Drill Span Name

Under **List of Back Drill Spans**, commands for **Back Drill Oversize** and **Back Drill Stub Length** have been added. Commands supported here too; **Must Cut** will return the last electrical layer to be drilled, and **Must not cut** will return the first electrical layer with a connection to the via after Back Drilling has been performed. Also supported, **Back Drill Span Name**, **Start Layer**, **Start Layer Number**, **Stop Layer** and **Stop Layer Number**.



Additional commands are available when using:

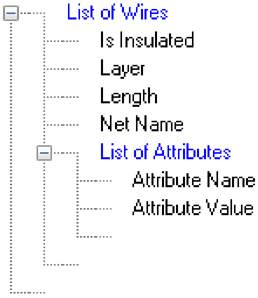
On a via - Is Back Drilled, Top Back Drill Span Name, Bottom Back Drill Span Name.

On a layer - Layer Depth.

On a Layer Span - Span Depth.

New command – List of Attributes available in List of Wires

With the introduction of **Attributes** for **Wires**, you can now extract the attributes and values using **List of Attributes** from within a **List of Wires**.



This command was back-fitted to 10.5

New command - Is Any Instance Fitted

A new command has been added **Is Any Instance Fitted** and can be used in a Schematic or PCB design report. Use within a **List of Variants** within a **List of Components**. This is best used in conjunction with the **If** command to test if any of the variant instances of the current Component are fitted in the current variant.

This varies from the **Is Fitted** command that is available to just check if the current Component instance is fitted in the current variant.

This command was back-fitted to 10.5

New command - Attribute In Part Library

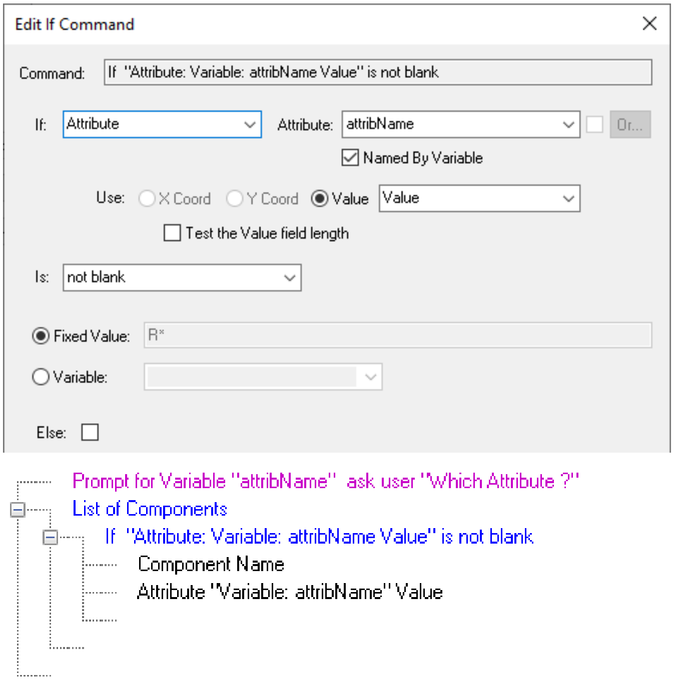
A new command **Attribute In Part Library** and can be used in a Schematic or PCB design report. Use on a Part, Component, Component Group or Associated Part to report the value of a named attribute on the latest Part in the library (or Vault).

This was required so that obsolete parts could be marked in the library (or Vault) and then a report created of which parts in a design are obsolete without having to reload all the parts. If the part is added to the design from the vault, it will get the value of the named attribute on the latest version of the part in the vault.

This feature was back-fitted to 10.5

Define the attribute name in an attribute command using a variable

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



This feature was back-fitted to 10.5.

New command - Has Track Sturs

A new command **Has Track Spurs** is now available to be used in a PCB report within **List of Differential Pairs**, **List of Differential Pair Chains** and also in **List of Pin Pairs** when used within the previously mentioned two lists. This reports if the pair, chain or one side of the pair or chain have track stubs in the path from start node to end node. This can be used in a **Variable** to compare against **Allow Track Spurs** to report an error.

Scripting Changes

Scripting changes are only outlined below. More syntactical details are available in the Online Help under Scripting.

Folders object

The *Folders* object provides access to the various sets of folder information available in the application through the **Setup Folders** dialog.

This allows you to add and remove folders across all possible categories, potentially allowing a script to be written to provide a full "change workspace" capability by programmatically resetting all folders.

New property "DeviceString" in a CamPlot object

There is a new property **DeviceString** in a CamPlot object. This allows access to the printer name of a plot.

This feature was back-fitted to 10.5

New method "ActivateDesign" in the Application object

There is a new method **ActivateDesign** in the Application object. This allows an already open design to be activated (brought to the front).

This feature was back-fitted to 10.5

New Scripting Commands for Export of ODB++ and DXF

New scripting commands are available for export of ODB++ and DXF.

This is an extension to the existing command in Scripting, which previously only allowed you to output plots for the 'normal' output devices (Gerber etc.). It now allows any 'device' that can be used in a plot on the CAM Plot dialog; DXF, Excellon, GenCAD, Gerber, IDF, IPC-2581, LPKF, ODB++, PDF, Reports, Pen, STEP, SVG, and Windows.

New Scripting Command for LoadTechnology

A new LoadTechnology function is now available for a Document object. Calling this function with a full path to a technology file name will load that technology into the design. If you wish to load part of a Technology file, you can do this now but you will have to save a partial Technology file and load the whole file in order to achieve this.

This is a simple VBScript example of how you could use this new function:

```
Set Design= ActiveDocument
if Design.LoadTechnology("f:\docs\data\technology\6 Layer.ptf")
then
                Message "Success"
else
                Message "Failure"
end if
```

Call the function with a full path name to the required Technology file and it will be loaded into the design. You can do this with a full or partial Technology. You can to use this for example to load just CAM/Plot settings by first saving the required section out as a partial technology file. Help has been updated to show the scope of this change.

Changes to Import Alien Design Files and STEP Model Positions

When importing Altium designs as ASCII files, Pulsonix now reads the STEP model name, offsets and rotations and places them into the three STEP attributes on the Footprint within Pulsonix.

Note, there have been some examples supplied that have no rational logic for model rotations and therefore have to be manually adjusted once in Pulsonix. However, these are rare and usually only one model in a design.

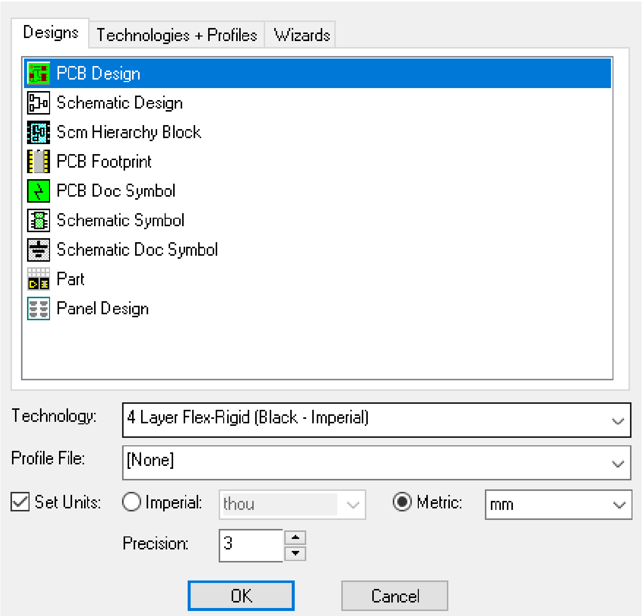
Loading Partial Colour Files

You can now load a cut-down version of the **Colours** file for both SCM and PCB. This should only be done if you know what you're doing. Further information and formatting details can be found on the Help page under the **Index** heading **Load Colours**

New PCB Design Dialog – Set Design Units

From the **File** menu and **New Designs** tab and **PCB Design**, you can now choose to overwrite the **Units** defined in the **Technology File** selected.

By selecting the **Set Units** check box, you can then select between **Imperial** and **Metric**, choose the **unit type** and set the unit **Precision**.



Changes to Default Supplied Files

Technology Files

All Schematic and PCB Technology files have been updated to reflect the new rule sets in V11.0.

Libraries

The Schematic Doc Symbol library now includes example of new Doc Symbol types for **Net Label**, and **Mounting Hole**. Existing Doc Symbols have been redefined to give them the correct category used for searching within the **Insert Signal Reference** dialog.