Pulsonix Design System
V12.0 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2022
Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

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

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

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

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

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

Microsoft, Windows and Windows NT are either registered trademarks or trademarks of Microsoft Corporation. All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 23/05/22 iss 1

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

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

Contents

CONTENTS 3
VERSION 12.0 UPDATE SUPPLEMENT 6
Installing the New Version of Pulsonix 6
Licensing 6
Version 12.0 Update Notes 6
NEW IN VERSION 12.0 7
64-Bit Operation 7
Dark Mode UI 7
Switching to Dark Mode 8
Using 'Light' Mode 9
Using Legacy Icons 9
Changes to Customise option 10
Show/Hide Menu Icons 10
Toolbar Icon Scale Slider 10
Changes to 3D and STEP 11
STEP Preview Multi-Threading 11
New 3D Design menu 11
3D View tab is now named 3D Packages 11
Mouse Button Change for Rotate in Preview 12
3D Settings Output – Output folder and Design Name 12
The 3D Settings dialog has been reorganised 12
3D Settings - Silkscreen Layer Class 12
3D Settings Output - Z-fighting items 13
STEP Preview - Multiple boards in one design 14
Inspector Bar – Board Group 18
Changes to Library functionality 18
Part Editor - 'Map Multiple PCB Pins' 18
Change Part Type of an Associated Part 19
Part Edit – Force Display of Attributes 19
Multi-Threading for Find Part in Library 20
Multi-Threading for Find on the Insert Component option 20
Multi-Threading for Library Indexing 20
Change Part Dialog - Same Symbol filter 21
Blue and underlined hyperlinks in Part Browser & Insert Component dialog 21
New Mode for Type Rotate 22
Move item(s) Onto Grid option 22
Performance Improvement – Interrupt for design clearance 22
New highlight Colour for Components Not Fitted in any Variant 23
Variant Manager – Delete Unused Components or Unfit option 23
Reload Block - Preserve block instance variant 23
New Design Setting to allow Pad Exceptions to 'Move with Buried Component' 24
Properties - Override Land Suppression on individual Pads and Vias 24
Deselect on all pages option in Schematics 24
New Pour All Templates Command 25
New Insert commands available – Ellipse, Spline, Wave 25
Splines 26
Waves 28

Ellipse 29
Changes to Existing Spiral Shape Algorithm 30
Insert Bus Route Changes 31
Changes to Construction Lines 32
Text Search in Find Bar 32
Change 'open' Copper to Track and vice versa 33
Area Mounting Hole Keep Out/In 33
Remove from Group when multiple items are selected 34
Match Rotation/Mirror of previous Component from Bin 34
Auto Rename - Padded with zeros 34
New 'Replicate Shape As' feature 35
Star Point Design Settings Defaults Changed 35
Highlight Net includes Star Point 35
Option to Synchronise the Document Title & Subject 36
Option to Suppress UTF-8 BOM used in PLM and Scripting 36
Toggle Layers 37
Redesigned Toggle Layer dialog 37
Layers Bar – Create Layer Toggle From Selected Layers 38
Technology Dialog Changes 39
Grids Page - Apply Origin Settings To All Grids 39
Spacing rules - Design level - Pad - Suppressed pads - Additional spacing for Drill Holes 40
Component Colour Rules 40
Minimum Pad Land for Mounting Holes 41
No Same Component Pad to Pad Errors extended to include Mounting Holes 41
Suppress Pads on Outer Layers 42
Anti Pad Rules 42
Associate a layer with a different non-electrical class to Swap With 43
Match Pair Spacings for Diff Pairs, Signal Paths and Sub Nets 44
Copper Pour – Keep Touching Copper 44
Define Teardrop by Length 45
Loop Antennae Rule 46
Stub Routing Length Rule 47
Track Mitring Rule 48
Return Path Rule 48
Copper Check Rule 49
Minimum Text Size Rule 50
Min Solder Mask Width Rule 50
Solder Mask To Track Rule 51
SMD To Corner and SMD To Plane Rules 52
New System Attributes Available 53
Added Last Saved Author Attribute 53
Added Last Saved Year attribute 53
New in-build attribute of <Not Fitted In> added 53
Attribute Validation when adding a Component 53
Additional DRC Checks 54
Silkscreen Overlap Doc Shapes 54
Check for Modified Templates 54
DRC Dialog – Shows Errors Bar can now show Fully Expanded Bar 54
Apply Layout Pattern – Type Final Position 55
Editing Differential Paired Tracks (High-Speed option) 55
Design Calculators - Track Width Calculator Changes 55
Panel Editor Changes 56
Set PCB Panel Origin Command 56
Min Copper Spacing for Tab Routs and V-Scores 57

New option to Create Tab-Route Around PCB Avoids Existing Tab-Routs....58
CAM Plot Changes....58
PDF Export – Plot and Combine PCB and Schematic designs....58
PDF Export - Plot and Combine PCB and Schematic designs share Bookmarks....59
PDF Export - Combine plots by Group....60
PDF Export - Allow click on page link symbol to go to page....60
PDF Export - Selectable Hyperlinks....60
ODB++ Export Changes....61
Changes to IDF Output option....61
Option to not include Component Attributes....61
Remove Illegal Characters from Part Names....62
Change Units and Precision....62
Changes to Intelligent Gerber Import option....63
Vault Setup – Pre-check status for Vault Options....64
Report Maker Changes....64
Scripting Changes - New LoadTechnology Command....71
Network License Server (NLS) - Send Message option....71

Version 12.0 Update Supplement

Installing the New Version of Pulsonix

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

To install Pulsonix, double-click on the download executable and wait for a short time. 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 12.0 over your existing V11.0 installation. If upgrading from V11.0, you can install it alongside the older version if you prefer. In any case, you do not need to uninstall the old version first unless you wish to remove it from your hard drive.

64-bit Installation Folder

By default, Pulsonix V12 will be installed into the programs folder **C:\Program Files\Pulsonix12.0** rather than C:\Program Files (X86) which is where the 32-bit applications are installed.

Documentation Installation

The default installation locates all Pulsonix 'documents' (Master Libraries, Technology files etc.) is under user\documents\Pulsonix12.0 rather than being placed in public documents\PulsonixXX

Licensing

Version 12.0 requires a new license if you are a new user or upgrading from any older version of Pulsonix earlier than V11.0. 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 12.0 Update Notes

The **Version 12.0 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 V12.0 Updates Notes**. You can also find them on the Pulsonix web site. The set of notes on the web site will be more up-to-date than the installed version as we are constantly updating these.

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

New In Version 12.0

64-Bit Operation

Pulsonix has been redeveloped as a 64-bit application. As such, it will now be delivered as a 64-bit only product with no 32-bit application available. All functionality will be the same as previous. This means, if you have an older 32-bit Windows operating system, you will not be able to install or run Pulsonix 12.0

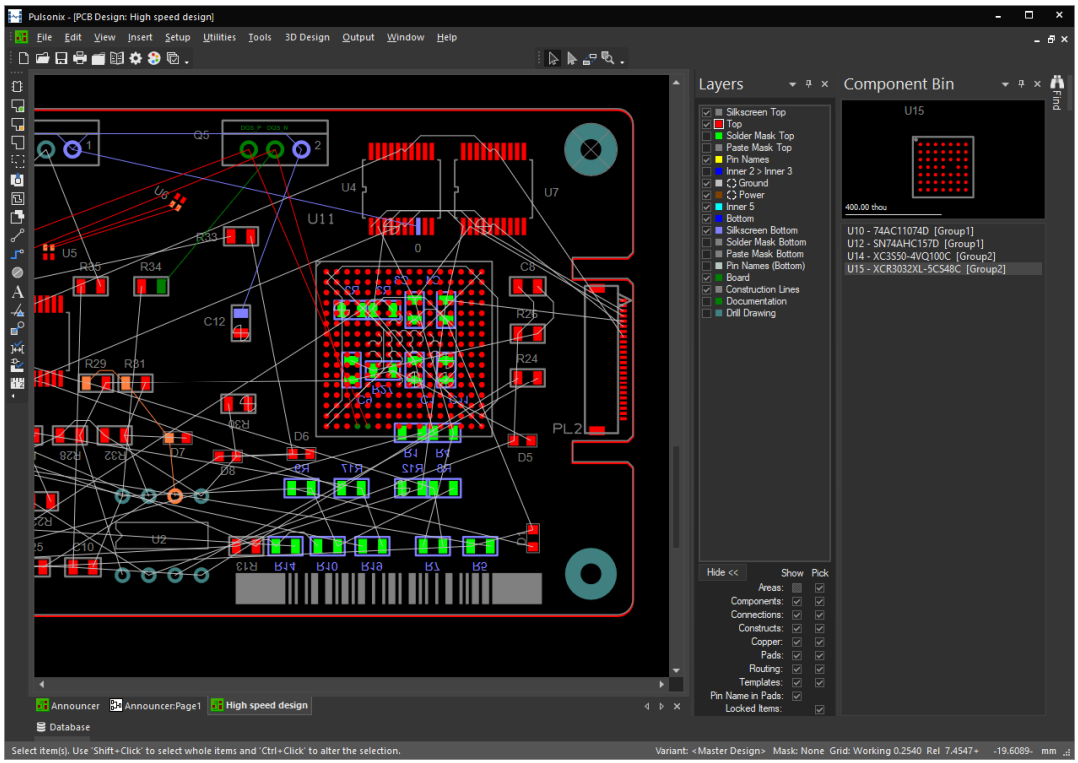
With 64-bit, the existing 32-bit address space memory limitation of 4GB has been increased, effectively removed, with 2TB+ now available. The advantage of this is a performance improvement for large designs.

NLS Floating license server

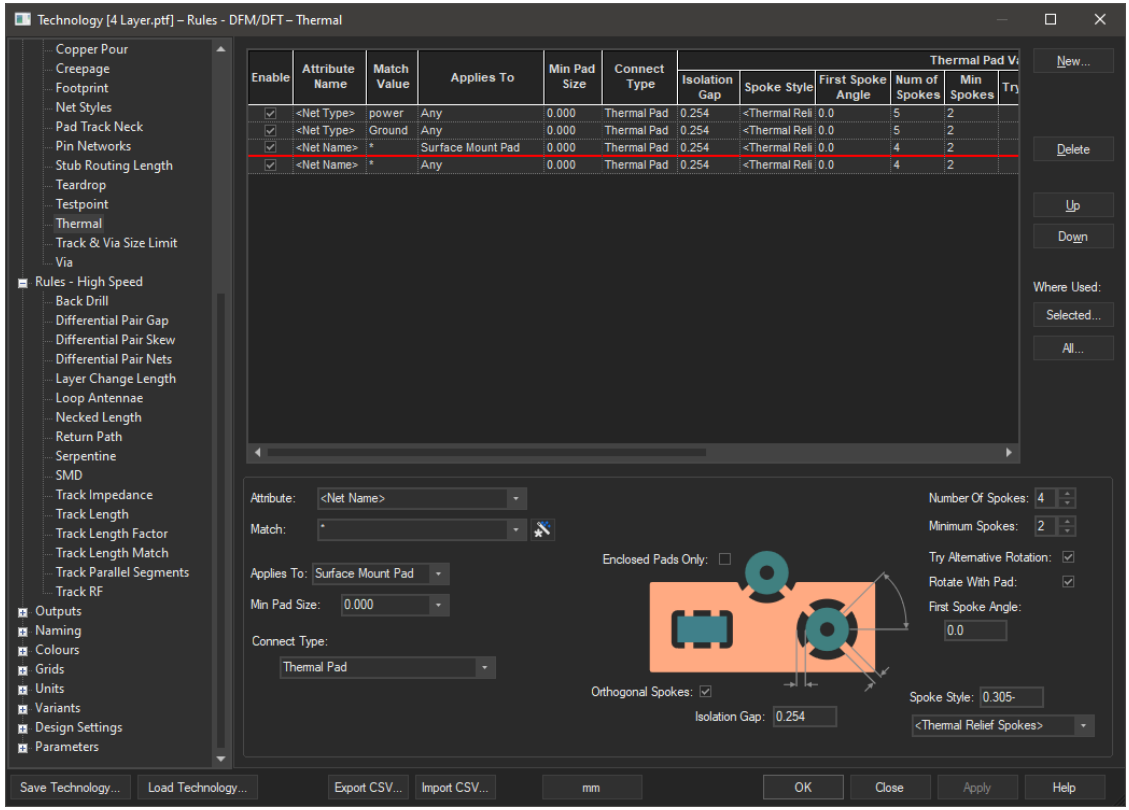
The NLS server program has been rebuilt as a 64-bit application but does not need updating for use with Version 12.0. Older versions of NLS for V11.0 and V10.5 will still work with the new V12.0 clients. A new server license will be required though to run V12.0

Dark Mode UI

A new alternative Dark UI theme is available for selection. This can be easily switchable between Light and Dark modes.



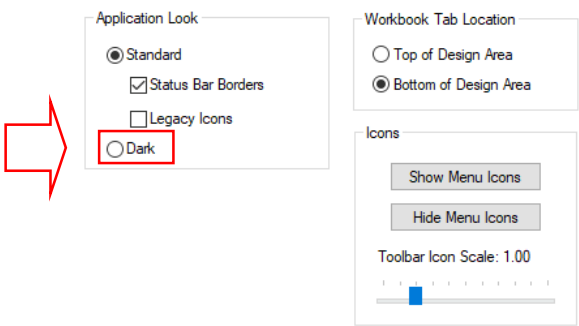
As well as the user interface, all dialogs will now be the Dark mode (if Dark mode is selected), for example, the **Technology** dialog:



Note, when in Dark mode, you cannot choose **Legacy** icons, see below, only the Dark mode icon set will be available.

Switching to Dark Mode

Dark mode is enabled using the **Customise** dialog and **Application Look** tab:



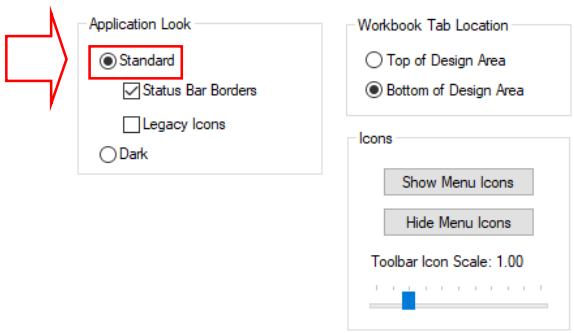
Select the **Dark** radio button and use **Apply Changes** to reconfigure Pulsonix to this mode. This not only enables the dark mode for all dialogs but also swaps out all toolbar icons to a set that is more compatible with dark mode.

Dark Mode and License Manager

When switching to Dark mode, you will also notice the **License Manager** adopts the same Dark style, this is automatic and cannot be switched separately.

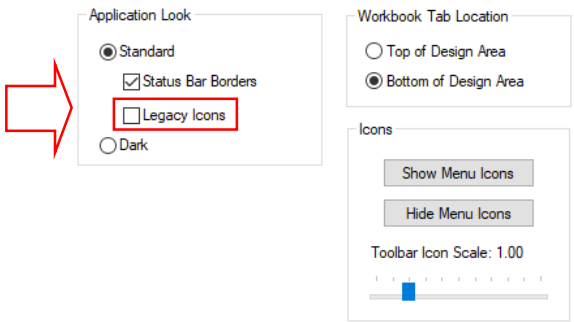
Using 'Light' Mode

When using 'light' mode (**Standard**) there is now also a new alternative icon set which compliments the dark icon set. This is a new icon set and is now the default installed set for use. It can be swapped to **Legacy Icons** if really required, see below.



Using Legacy Icons

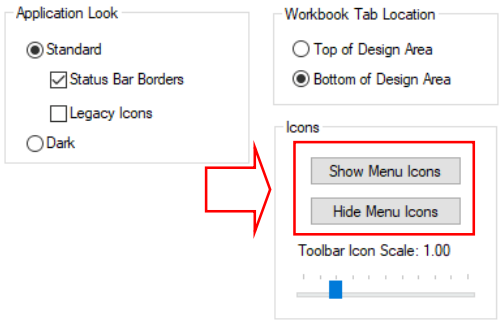
A check box has been added under the **Standard** look radio button which will set Pulsonix to use the original, coloured icon set (**Legacy Icons**). By default, the new 'light' set of icons are used in Pulsonix. The **Legacy Icons** feature is disabled for the dark mode interface where only the dark icon set can be used.



Changes to Customise option

Show/Hide Menu Icons

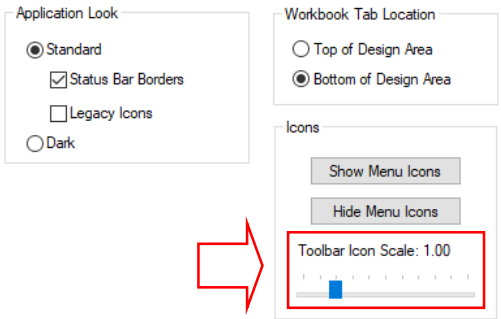
To aid menu decluttering, you can **Show** (or **Hide**) **Menu Icons** from with the **Customise** dialog and **Application Look** tab. By default, on a new installation, menu icons are shown.



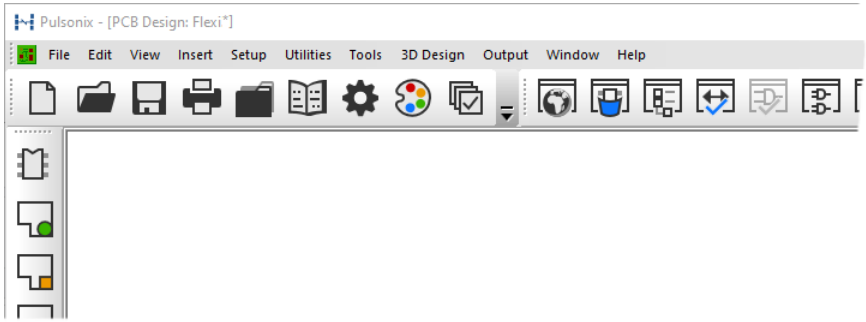
Toolbar Icon Scale Slider

The **Large Icons** switch in **Customise** and the **Options** tab has been removed and replaced with a slider feature. This can be used when running a 4K monitor and requiring slightly larger icons. Because the Pulsonix icons are SVG format underneath, they can be scaled and still look usable.

There is now a slider in the **Customise** dialog and **Application Look**. This will allow you to change the size of the toolbar icons displayed to your preference.



The icons below are shown at a scale of 2.00x



Changes to 3D and STEP

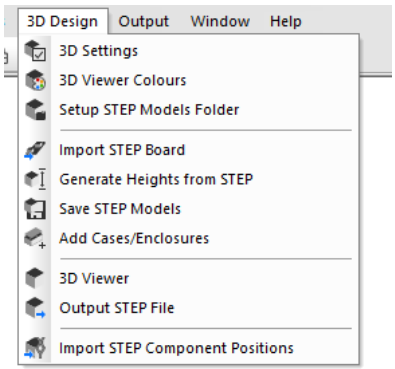
STEP Preview Multi-Threading

STEP file generation now uses multi-threading. It creates components in parallel, and also creates all items checked in **Advanced Settings** in the **3D View Settings** in parallel. Because of restrictions with the STEP 3D toolkit used in Pulsonix, multi-threading for this option is hard coded to 3 threads to protect it.

New 3D Design menu

A new **3D Design** menu has been added to bring most aspects of the 3D STEP functionality together in one location. The original **STEP Output** is still available on the **Output** menu in-case you have accessed it from a Macro and familiarity. This functionality and other STEP options are now only available on this new menu.

The **Edit** menu **Commands** have also been updated to reflect new options access.

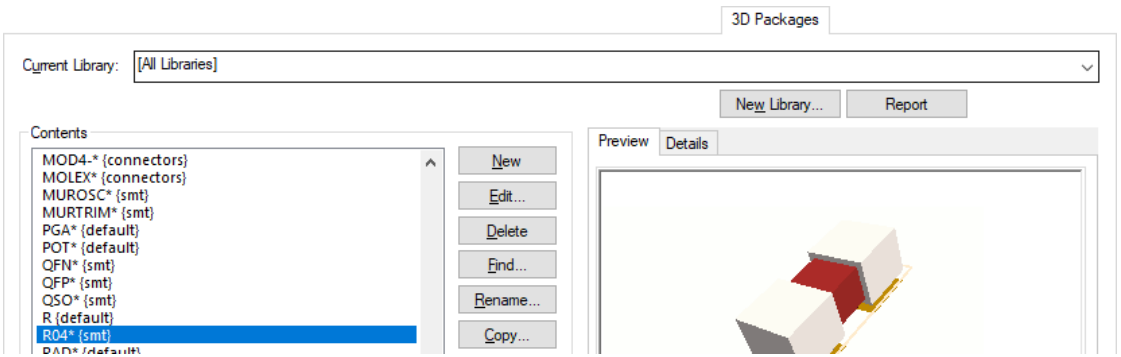


You will also notice that some options have been renamed to be more reflective of their use and for clarity. This menu currently contains **STEP** features. All other MCAD exports and features, such as for **DXF** and **IDF** are still on the **Output** menu and other respective menus.

This feature was back-fitted to V11.0 at patch 8466.

3D View tab is now named 3D Packages

The **3D View** tab on the Library Manager is now 3D Packages. This reflects is function and storage rather than its use.



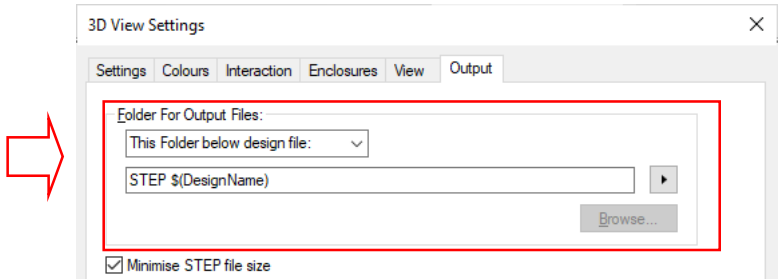
Mouse Button Change for Rotate in Preview

When in the STEP Preview, you can now press the **right mouse button** to **rotate** the board. Previously, this was Ctrl-right button.

3D Settings Output – Output folder and Design Name

The **3D Settings** dialog and **Output** tab has been changed to add the destination and formatting of the output files created when using the **3D Design** menu and **STEP Output** option.

This uses the standard file naming conventions used for other existing options, such as **CAM Plots** and **Reports**. You can choose between outputting to the design folder, defining a folder, defining a folder below the design file, and the general folder. When defining a folder, the arrow to the right of the control can be clicked to show a list of available tags to use. Click the browse button to search for and define a folder manually.



This feature was back-fitted to V11.0

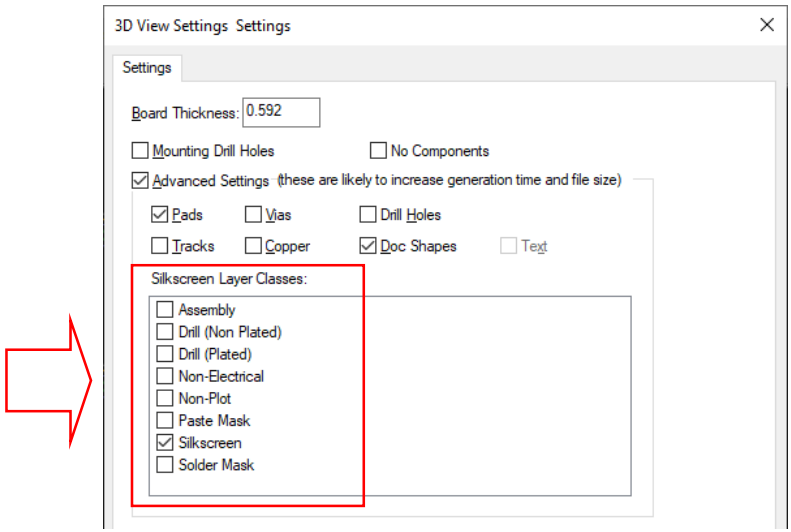
The 3D Settings dialog has been reorganised

Below is a summary of the rearranged controls on the **3D Settings** dialog. The changes are as follows:

- **Show Exploded View of Layer Stack** section has been moved from the **Settings** tab to the **View** tab.
- **Cross Probe** section has been moved from the **Settings** tab to the **Interaction** tab.
- **Place items directly on-board surface** and **Only when generating STEP output file** have been moved from the **Output** tab to the **Settings** tab.
- Added the option to define **Board (flexi) colour** on the **Colours** tab.

3D Settings - Silkscreen Layer Class

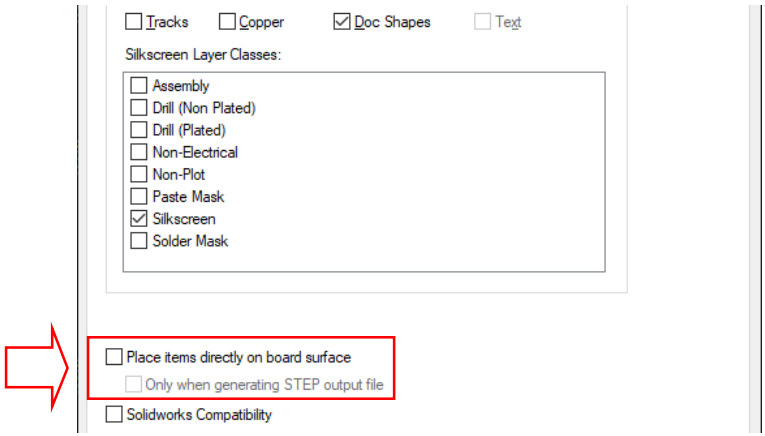
The **Silkscreen Layer Class** on the **Settings** page of the **3D Settings** dialog has been changed from a drop-down list (combo box) to a check list box that allows you to enable more than one class to be chosen to identify the layers for **Doc Shapes** and **Text**.



3D Settings Output - Z-fighting items

To avoid graphical issues (dithering, also known as Z-fighting) when drawing two or more items at the precise same location in the STEP viewer, 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 **Settings** tab of the **3D Settings** dialog allow you to do this:



The default setting for **Place items** check box is unchecked. Checking this box enables the next check box. This second check box 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.

This feature was back-fitted to V11.0

STEP Preview - Multiple boards in one design

The 3D Preview is now capable of viewing multiple boards. For this to operate, boards must be in the same PCB design, they cannot be separate designs.

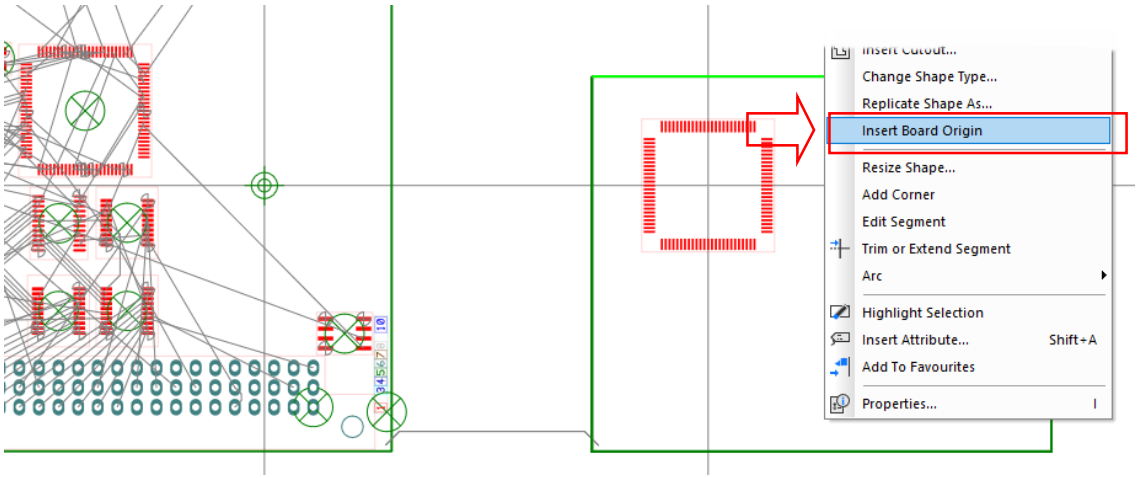
To facilitate this new operation, additional features have been added to Pulsonix. These are required for defining the Board Type, the point of movement on each board (Board Origin), Properties of a Board Origin from where the X,Y,Z parameters of the moving board are defined and additional parameters in the 3D Preview

Insert Board Origin

Insert Board Origin: A Board Origin or rather a pair of board origins, are used as the point of positioning from one board to another. The 'placed board' origin (the one moving), can contain Properties for **folding**, see below.

If there are only two Board Origins, these will be paired. If you have more than two boards that need to be folded, then add more pairs of Board Origins. In this case, you will need to name them so that the correct 'pair' are matched. For example, on three boards that should be folded together, you would have two pairs of Board Origins.

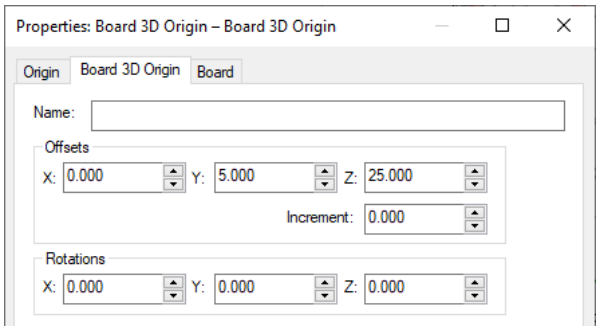
To access this new command, it is not on the main menu so right-click on the **Board Outline** and select **Insert Board Origin** from the context menu.



This allows you to place an origin marker that belongs to the board outline. Place one on each of your boards (optionally naming the origins to ensure that the correct ones are paired). These act as the points which are automatically brought together when the board is 'folded'.

In order to position the board, the Board Origin has a set of its own Properties. With the **Board Origin Properties** dialog**,** you can set the **name** (optional), set the XYZ **offsets**, set the XYZ **rotations**.

The **Increment** entry allows you to changes the 'step' that is applied when you click one of the spinners for X, Y or Z. This provides a more efficient way for you to state that you want to step in increments of say 0.5 instead of whatever value we choose. Most spinners used are for integers so it is easy to step up/down by 1, but these are floating point numbers so the preferred step could be just about any number.

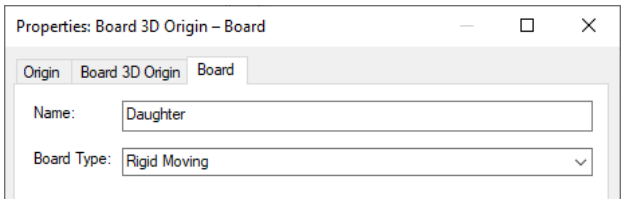


By specifying rotation and/or offset, this allows the moved board to be further positioned after it has been automatically aligned based on the pair of board origins. For example, to turn your secondary board over and 'hover' it above the main board, specify a **Z Offset** of say, 9mm and a **Y rotation** of 180.

Note, the Name used on this page is only the Board Origin Name, not the Board Name.

Board Properties for Board Outlines

There is now a **Board** properties page in **Properties** which can be accessed when a single board or multiple boards are selected. This page contains controls for **Board Name** and **Board Type**, taking values of Rigid Fixed, Rigid Moving, and Flex (although this one is not currently used anywhere).



The **Board Name** helps you identify it. This name is not related to the Board Origin.

The **Board Type** is used when moving/folding the board in the 3D Preview. If **Rigid Fixed**, it will not move and would be considered the 'master'. If **Rigid Moving**, it can be moved and the parameters on the Board 3D Origin page will be used.

3D Settings - Board Folding

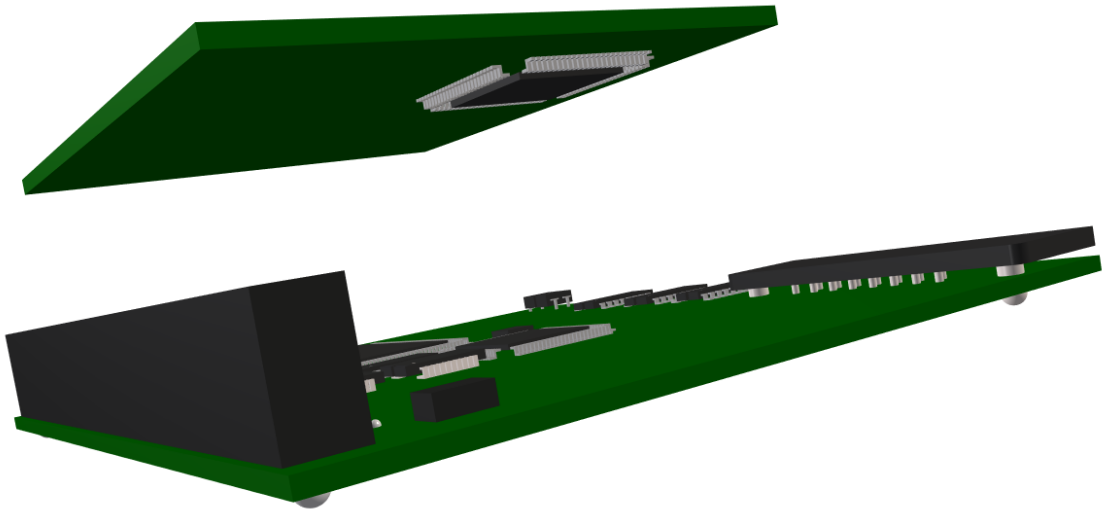
Once multiple boards have been created, Board Origin pairs defined, Board Types defined and Board 3D Origin folding parameters set (so the folding knows how much and where to position the board) you can now view the boards in the 3D Preview. Once in this editor, you can then perform the folding.

The new **Fold Board** command on the **Tools** menu is used to move a rigid Moving board to another defined position (from Board Origin and Board 3D Origin).

Selecting this tool causes the moving board(s) to be aligned/transformed as defined by the paired board origins. Everything inside the moved board (components, tracks, pads, etc.) will be moved in the same way so they move with the board. Invoking Fold again will toggle and 'unfold' the moved board back to the initial state as generated from the 2D PCB.

Note: when folded, components currently cannot be moved, this can only be done when in the unfolded state.

Also, the design will not be displayed folded if Exploded view is selected.



3D Settings – Clash Detection

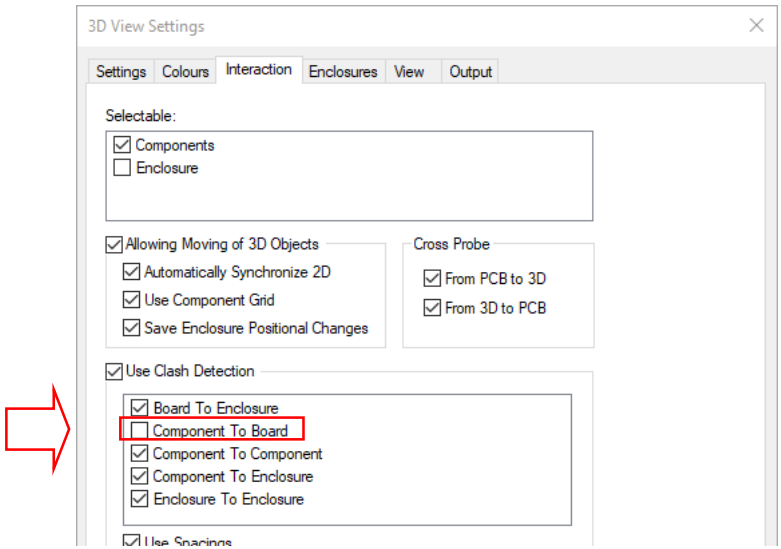
Clash Detection will work as before, and flag any clashes where components violate each other. However, if you have situations where you explicitly want items to touch (e.g., where a plug on the moving board fits into a socket on the static board) you can add a new attribute **3DIgnoreClash.** This should be added to one of the components that will be violated with a value of the component name of the other component (or a valid component name-range including comma separated names).

When the **Clash Detection** option is then run, the component will be ignored.

STEP Component to Board clashing

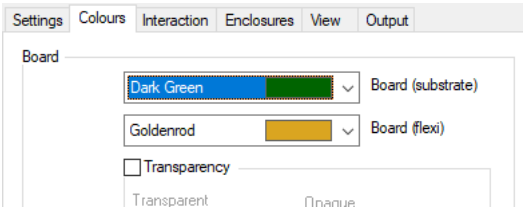
The **Detect Clashes** option in the **STEP 3D Preview** now can perform **Component to Board** clashes. This clash detection is required for checking clashes between the static and moving boards when using the **Fold** option. When folded, there is a possibility that the static board will clash with the moving boards' components, and vice versa.

To enable this clash detection, check the **Component To Board** check box in the **Clash Detection** section of the **3D Settings Interaction** page.



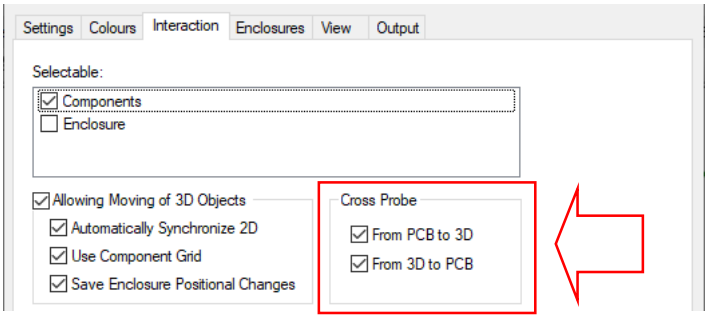
3D Settings - Board Differentiation Colours

Colours Board (flexi) - option to define **Board (flexi) colour** on the **Colours** tab, the **board type** must be set to **Flexi** for this to be visible.



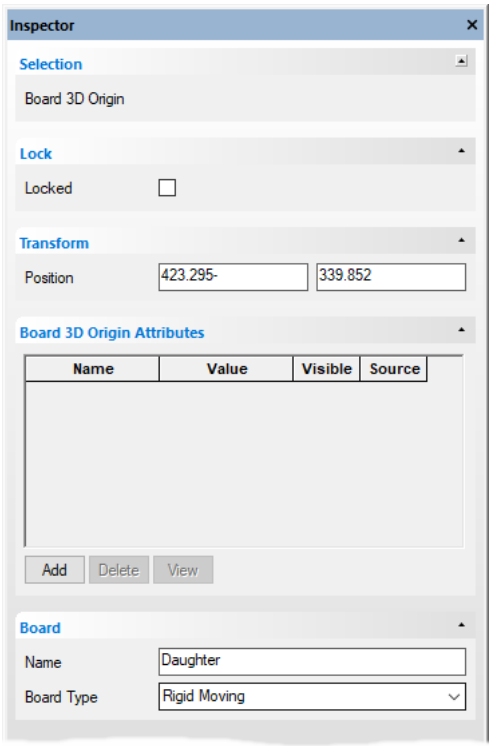
3D Settings – Cross Probe

Two new check boxes on the **3D Settings** and **Interaction** page have been added. These will enable **Cross Probe** in each direction between 2D PCB and 3D views. Clicking on items in one will select them in the other.



Inspector Bar – Board Group

A **Board Group** has been added to the **Inspector Bar**.



Limitations

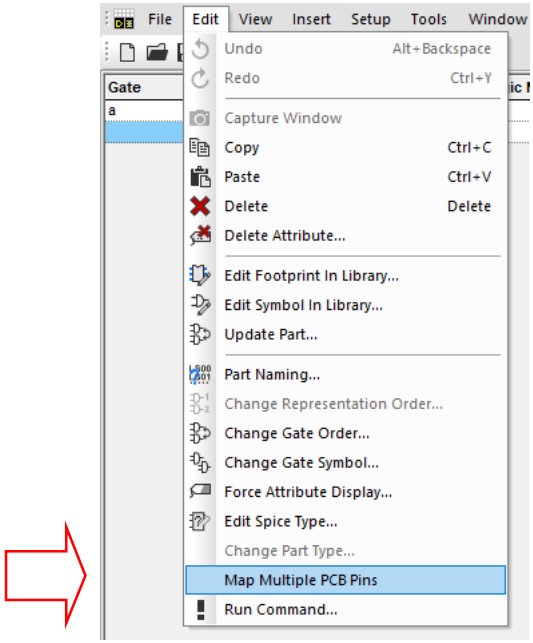
Cannot import a STEP assembly on another board type for ‘folding’.

No connection or bended flex is shown between boards and there is no animation when folding.

Changes to Library functionality

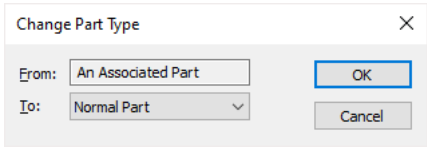
Part Editor - 'Map Multiple PCB Pins'

Within the **Parts Editor**, when editing a Part, the function **Map Multiple PCB Pins** is now also available on **Edit** menu. This is in addition to it being available on the context menu on the selected **Pin Names** cell on the **Gates** tab.



Change Part Type of an Associated Part

In the **Part Editor** you can now change an **Associated Part** to a **Normal Part**. Select the **Change Part Type** option from the **Edit** menu whilst editing an **Associated Part**.

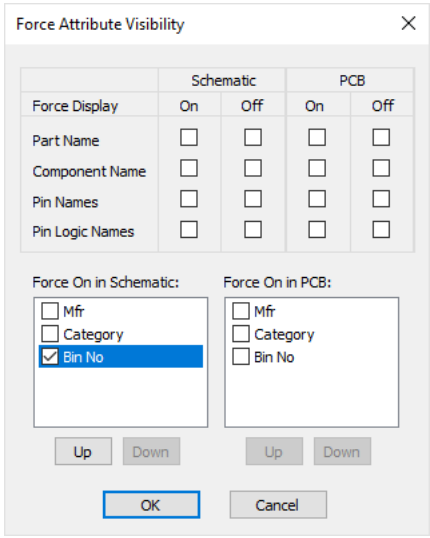


Note, this option already existed for changing a **Connector Part** to a **Normal Part** but has not been expanded to include **Associated Parts**.

Part Edit – Force Display of Attributes

When **editing a Part**, the **Force Attribute Displayed** dialog (from **Edit** menu) now includes two lists for you to define which attributes you want to force displayed when adding the Part to a Schematic or PCB design. This feature is particularly useful if you have imported a design from another product and wish to force specific attribute fields on.

If the attribute is not already on the symbol, the attribute positions will be added underneath each other in the order from the list. The order of the list can be adjusted using the **Up** and **Down** buttons under the **Force On** name column.



The **Force On** list is populated with attribute names that have already been used on the **Parts and Attributes** page of the **Parts Editor** for the Part being edited.

	Attributes	
Part Name	**Manually Name Parts**	A40MX02-PL44
Attributes	**Mfr**	Actel
	Category	IC/FPGA/MX
	Bin No	X344211

This feature was back-fitted to V11.0 and V10.5

Multi-Threading for Find Part in Library

The **Find Part** option on the **Parts Library** tab has been speeded up with the use of Multi-threading. This is only available when finding Parts in all libraries (not for single library find).

Multi-Threading for Find on the Insert Component option

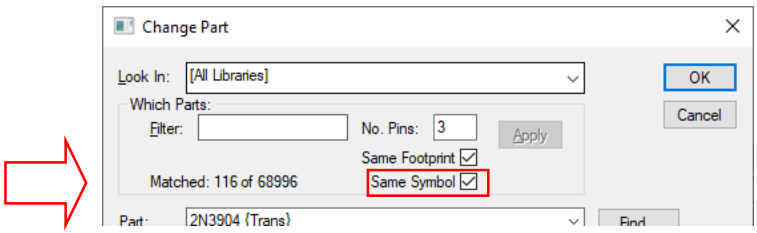
The of Multi-threading has been added to the **Insert Component** dialog**, Find** option. This will speed up the searching but is only available for all Parts and not single libraries.

Multi-Threading for Library Indexing

Multi-threading can now be used for library indexing. This is only available when indexing all libraries and not for single library indexing. This feature is available when multi-threading is enabled in the **Options** dialog and **General** page.

Change Part Dialog - Same Symbol filter

A **Same Symbol** filter has been added to the **Change Part** dialog when in schematic designs. The check box is used to limit the matching Parts to those that have the same symbol as the Part you are changing. This option is only available in schematic designs, and for Parts with a single gate.

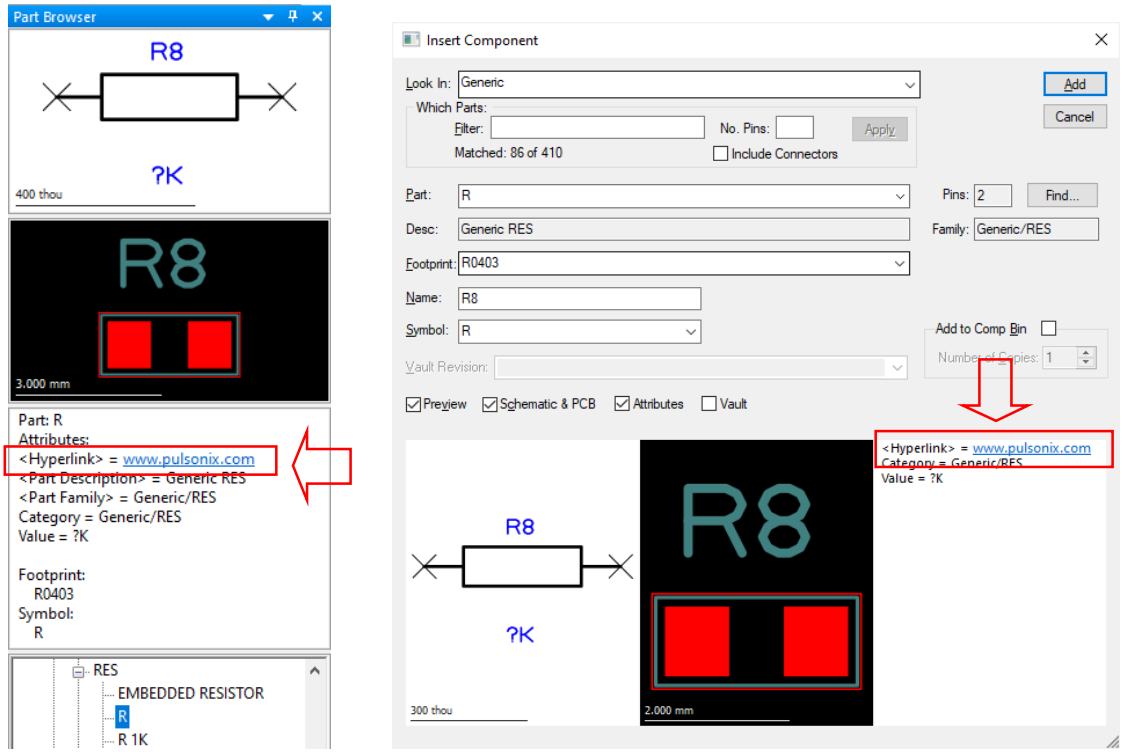


The symbol name has been added to the Part library index if the Part only has one symbol. When filtering by symbol in the **Change Part** dialog, if the symbol field is not currently in the library index for the selected library or libraries, it will be rebuilt.

This feature was back-fitted to V11.0

Blue and underlined hyperlinks in Part Browser & Insert Component dialog

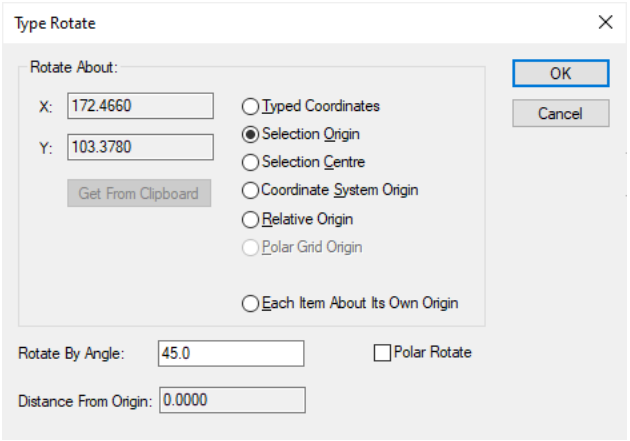
Hyperlink attributes in the **Part Browser** and **Insert Component** dialog attribute details pane are now shown in the standard link format (blue and underlined) to make it easier to identify them as clickable links.



New Mode for Type Rotate

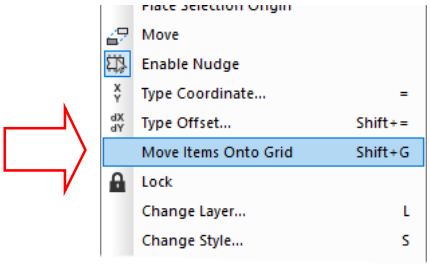
The **Type Rotate** dialog now has a new **Rotate About** option named **Each Item About Its Own Origin**. This will rotate each of the selected items (apart from tracks and connections) about their individual origins rather than rotating the selection as a whole.

This is useful where footprints in the library have been changed and they should be rotated in the design after an update.



Move item(s) Onto Grid option

There is now an option on the context menu for selected items to move onto the grid, named **Move Items Onto Grid**. Each selected item will move to the closest point of their appropriate grid (the current **Interactive Grid**).



If a track is selected it will not be gridded unless only a single track segment is selected. The track segment will move onto the grid based on its orientation.

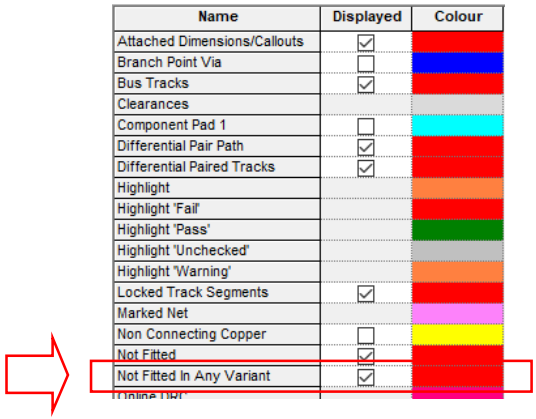
Performance Improvement – Interrupt for design clearance

The performance has been improved when moving very large components with Online DRC and Draw Clearance enabled, and Show Weld Spots. The time taken to calculate their design clearances (if enabled) has been improved.

This feature was back-fitted to V11.0

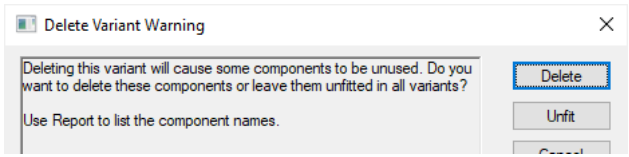
New highlight Colour for Components Not Fitted in any Variant

Similar to the **Not Fitted** highlight colour. The **Not Fitted In Any Variant** option will have a different colour for a component which does not have a variant assigned to it.



Variant Manager – Delete Unused Components or Unfit option

Within the **Variants Manager** in the **Technology**, using the **Delete** function now allows you to decide to delete unused components or just unfit them. A warning dialog appears showing three buttons.



Delete, will delete the components from the design,

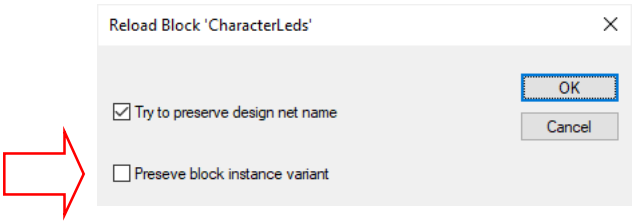
Unfit will unfit these components from all variants, but will remain in the design.

Cancel will cancel the delete and return you to the **Variant Manager** dialog.

Report will generate a list of components that are in the soon to be deleted variant and not in any other variant.

Reload Block - Preserve block instance variant

When Reloading Blocks, there is now a special option named **Preserve block instance variant** which when selected, preserves the Fitted/Unfitted status of Components in the block.

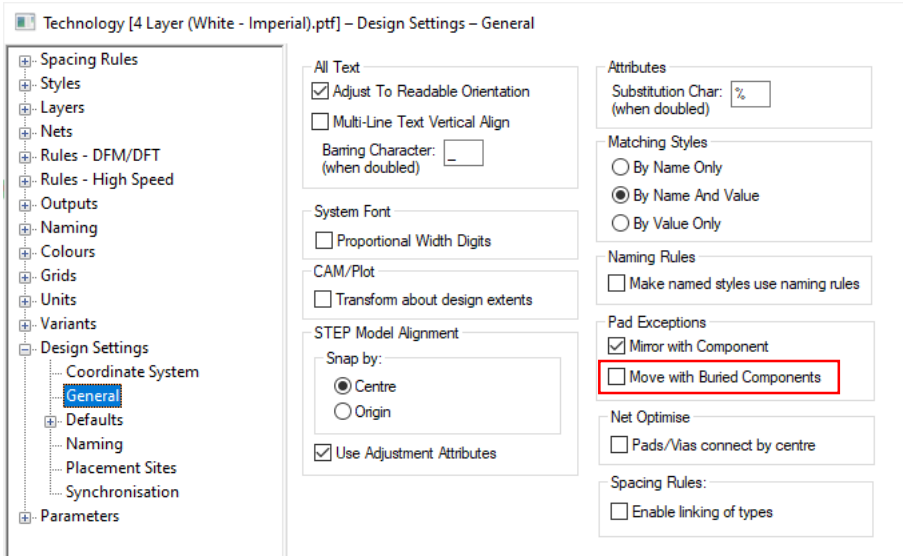


Note: This switch is only visible and only applies provided ALL Components in the block have the SAME status.

New Design Setting to allow Pad Exceptions to 'Move with Buried Component'

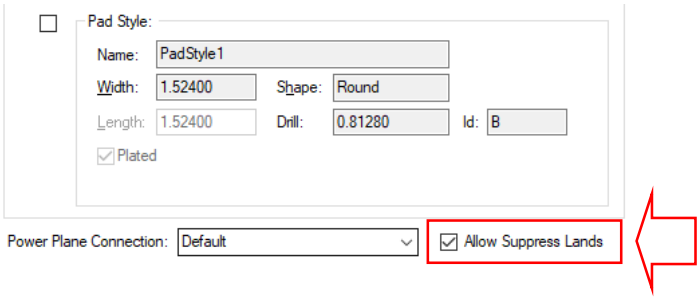
A **Design Setting** on the **General** page to allow Pad Exceptions to **Move with Buried Component** has been added.

This allows the **Pad Exceptions** to follow the buried component as it moves down the layer stack. For example, a Solder Paste exception on the Top Solder Paste layer would appear on the Solder Paste layer associated with the top facing layer a buried component is placed on.



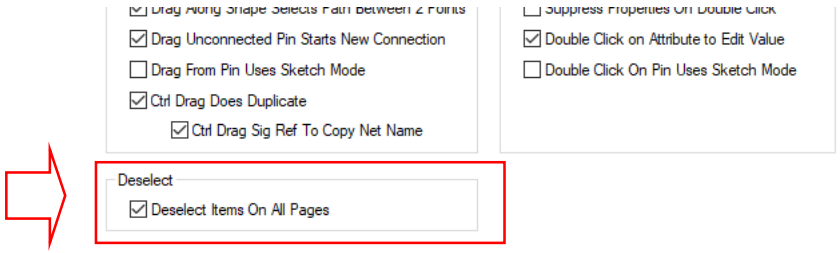
Properties - Override Land Suppression on individual Pads and Vias

In **Properties** on **Pads, Mounting Holes**, **Vias** or **Micro-Vias**, unchecking the **Allow Suppress Land** switch overrides the **Suppress Unconnected Land** flag on the layer (**Suppress Unconnected Land** is defined on the **Layer** page in the **Technology**).



Deselect on all pages option in Schematics

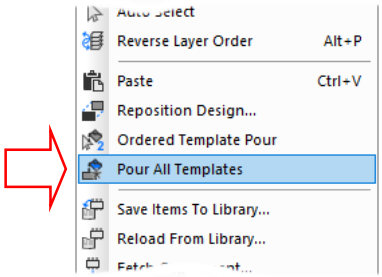
There is a new setting in **Options** – **Select,** under **Deselect** for **Deselect Items On All pages**. In a Schematic design, selecting this option changes the **deselect all** behaviour to deselect everything on all pages of the design.



New Pour All Templates Command

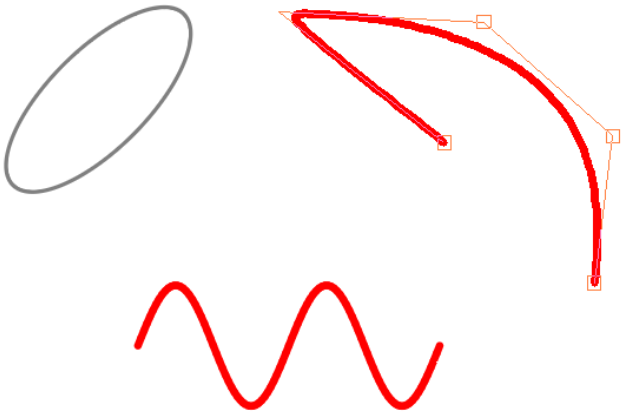
A new **Pour All Templates** command has been added. This is available on the **Utilities** menu. Selecting this option will pour copper into all templates on all layers using the pour order if defined.

This command is also available on context menu with nothing selected in the design.



New Insert commands available – Ellipse, Spline, Wave

New commands have been added to the **Insert** menu and various insert commands to add **Ellipse**, **Spline** and **Wave** shapes.



These shapes can be added to the following:

Board	Ellipse		
Copper		Spline	Wave
Shape	Ellipse	Spline	
Area	Ellipse		
Construction Line	Ellipse	Spline	Wave
Track		Spline	Wave

Splines

After selecting **Insert Spline** *shape* from the **Insert** menu, you draw the start and end point of the shape. The shape Control Points are then added; as many as required. The shape created is then save in the design as a controlled shape, meaning it retains the control Point handles for future editing.

Control Points are used to bend the curve towards it.

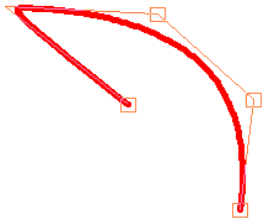
Control Point handles are available when adding or editing splines.

Single press a position on the design to place the start of the spline.

Single press another position on the design the place the end of the spline.

Begin adding **Control Points** by single pressing onto the design - the spline shape that fits through the points will be generated dynamically to show what the finished shape will look like.

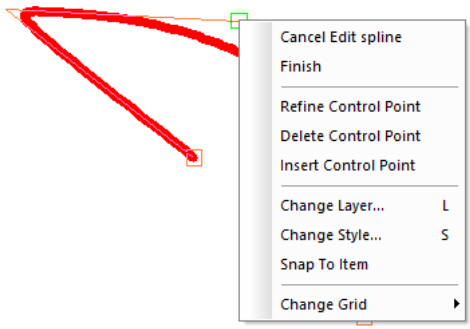
The order of the **Control Points** can be reversed during this stage if they are not in the expected order. Reverse the control points using the **Reverse Inner Control Points** command from the context menu.



Double click or use the **Finish Spline** command from the context menu to stop placing Control Points and enter the edit state.

If you edit a spline and single click on a Control Point (the small square box), from the context menu, you can **Refine the Control Point** to a different location, **Delete the Control Point** or **Insert Control Point**.

To use **Delete** or **Insert**, edit the shape, then left click on the small square you wish to edit and right click to reveal the context menu. Inserted Control Points are added after the one you have selected.



As an alternative, you can add a **Construction Line Spline** to the design and use **Follow Construction Line** from the context menu when adding a new shape, a **Track** for example.

Merging Splines

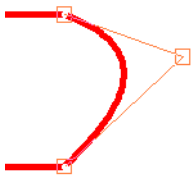
As with all controlled shapes (splines, waves, serpentines), you cannot use merge on them. If you wish to connect two splines, use a short track segment, and then overlay the two ends of the splines to create a zero sized track. Ideally, you would create one spline of the full shape required.

Spline Segment

With a track segment selected, from the context menu you can select **Spline Segment**. This will split a track and insert a spline using the start and end points of the selected segment.

Converting a Track Segment to a Spline

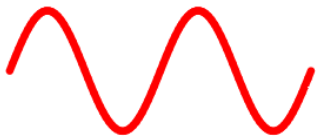
With a **single track** segment selected, the **Spline Segment** command on the context menu can be used to turn that segment into a spline. The start and end points of the spline will already be placed at the start and end points of the selected segment.



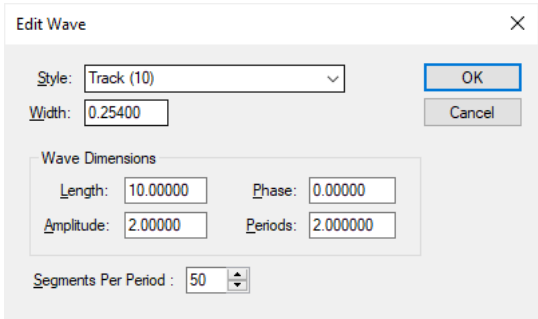
Remove Spline Shape

For a select a spline shape, from the context menu you can select, **Remove Spline Shape**. This command removes the spline-controlled shape and reverts the shape to a straight line between the start and end points. Once this has been done, it cannot be made back into a Spline (other than if **Undo** is available) without recreating it.

Waves

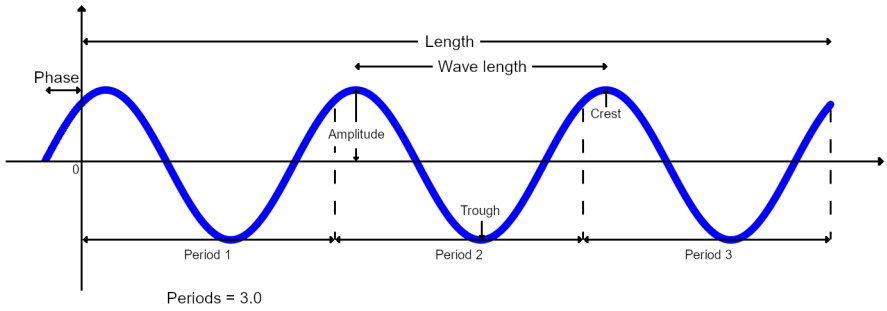


When adding or editing a Wave shape, a dialog is displayed from which to choose the wave characteristics:



Select the required parameters and then press OK. The wave shape will appear on the cursor and you can place the wave where you want. If the wave does not look correct, you can cancel the operation. This will return you to the dialog and you can adjust the parameters. When placing a Track, you can use the **auto weld** feature to connect the wave to a pad or via as you place it. You can **Rotate** or **Mirror** the wave as you place it.

Wave Dimensions



The **Length** is the total width of the wave shape.

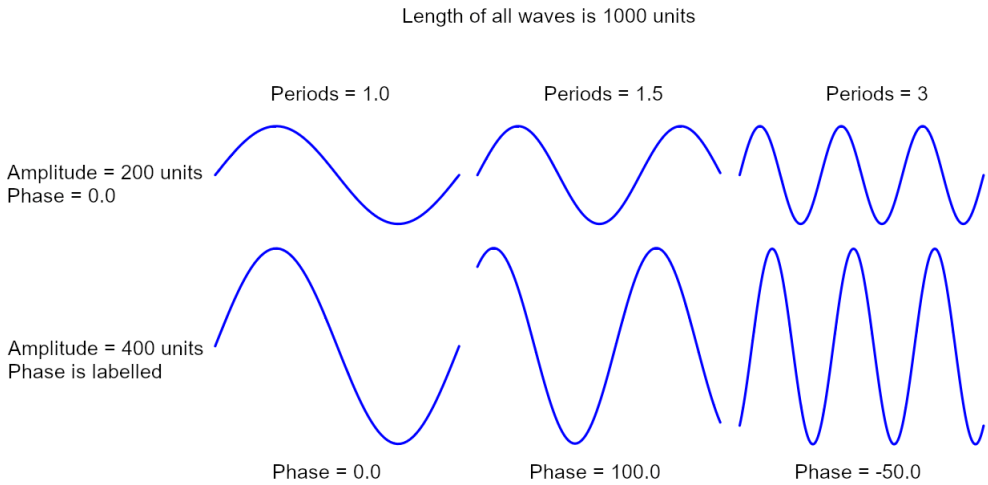
The **Phase** is the offset which is used before the wave begins. This can be used to create cosine waves by using a **Phase** which is exactly half of the *wave length* where *wave length* = **Length** / **Periods**.

The **Amplitude** is the height of each crest/trough from the centre line of the wave *(y = 0)*.

The **Periods** defines how many periods of the wave are fitted into the total **Length** of the wave.

Segments Per Period – this alters the accuracy of the wave. A higher value will create a more accurate wave by increasing the number of straight segments in each period.

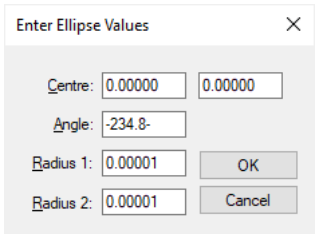
Below are some examples showing how wave forms are changed by altering the available parameters:



Ellipse

From the **Insert** menu, select *Board*, *Shape*, *Area* or *Construction Line*, and the **Ellipse** option.

At any point during the creation or editing of an ellipse, the **Enter Ellipse Parameters** command can be used from the context menu. This will display the centre position, angle, and two radii of the ellipse, as well as bypassing the interactive creation of the ellipse. You can edit these values for the ones you require.

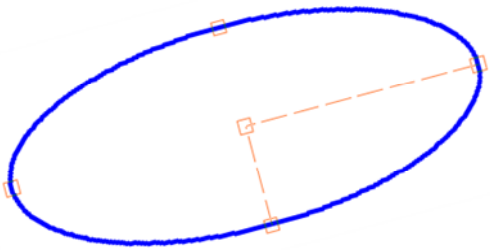


Start the Ellipse shape manually, select the start point in the design using the mouse.

- Move the cursor to define the size of the **First Radius** as well as the **Angle** of the ellipse. An ellipse can be defined from the centre or edge to edge by toggling the **Define from Centre** command in the context menu.

- You can move the cursor in any direction but will be constrained to 45-degree angle steps unless the **Free Movement** command is toggled in the context menu.
- Single press to finish defining the **First Radius** and the **Angle**.
- The **Second Radius** of the ellipse will then become active and allow changes. Again, this is done moving the cursor. The radius will be equal to the distance from the centre of the ellipse to the cursor.
- Press the mouse to finish defining the **Second Radius**.

After the two radii and the angle have been set, the ellipse can be edited. Four handles will appear on the perimeter of the ellipse, which can be dragged with the cursor to change the corresponding radius. The centre handle can be used to move the ellipse while in the edit state.



A handle may be selected by selecting it with the mouse. A handle will become highlighted, along with the one opposite, to indicate that it is selected. With a handle selected, the commands **Enter Radius and Enter Diameter** will be available from the context menu and can be used to manually enter the size of the radius.

The angle of the ellipse can be modified in the edit state by using either the **Rotate About** or **Enter Angle** commands. The **Rotate About** command will cause the ellipse to rotate towards the cursor similar to how it does when first creating the ellipse.

The two radii and the angle are displayed in the **Status Bar** to show the ellipse's dimensions during creation.

Changes to Existing Spiral Shape Algorithm

The existing **Spiral shape** has been changed with a new algorithm for calculating the spiral. Previously, the spiral was generated using approximated ‘half’ circles that offset each other. The use of the new Archimedean algorithm generates a more accurate and smoother spiral.

If you wish to use this new algorithm, then you will need to remove, then recreate the existing spiral shape. The shape is not regenerated automatically. Unlike the existing shape though, the new shape style will be retained and can be re-edited afterwards. The old shape was a one-off and not editable.

Insert Bus Route Changes

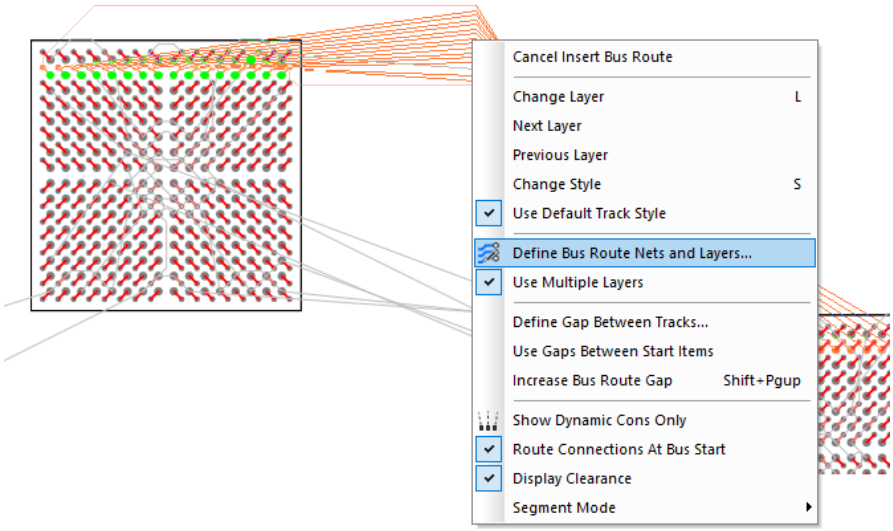
There are two changes to make bus routing more efficient:

The **Define Bus Route Nets** dialog will now change to **Define Bus Route Nets and Layers** when **Use Multiple Layers** is selected. The dialog then includes a **Layers** column so that the layer for the track can be defined.

Insert Bus Route will now make use of **Breakout patterns** on pads.

Layers column in Define Bus Route Nets dialog

With the **Use Multiple Layers** selected, when using the **Define Bus Route Nets and Layers** dialog, either to choose the nets to be included in the Bus Route or after picking the pads/connections to be included, there is now a **Layers** column in the dialog presented to define what layer you want each net to be routed on. This enables you to route sub-sets of related nets together on different layers which makes it easier to route out of BGA patterns and keeps the Bus Route Trunk width to a minimum.



The track segments will be added interactively on the multiple layers at the same time, online DRC will apply on all layers being used.

Finishing the Bus Route will result in separate Bus Routes being added on each layer involved. These can then be edited separately.

If the multi-layered bus routes are finished in space, they can be continued using the Insert Bus Route option by framing the dangling ends.

Use of Breakout Patterns on Insert Bus Route

When the **Insert Bus Route** option is selected, it will route the **gathering stage** (the initial phase of 'starting' the routing) of the chosen tracks in the bus route from tracks and vias created from the breakouts on the footprint pads (as created in the Footprint Library). It will also make use of breakouts when completing the Bus Route at the target end, thus making it easier to add Bus routes between BGA devices.

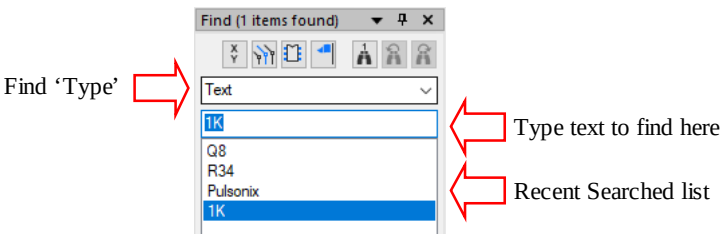
Changes to Construction Lines

With the addition of the new shape types to Construction Lines (Ellipse, Spline and Wave) (see above), this introduces new concepts to the option. Spline and Wave shapes are effectively 'local' Construction Lines and not just 'infinity' lines (although Circle Construction Lines technical are too).

With the addition of **Spline** and **Wave Construction Line shapes**, you can use this as any other Construction Line shape in that you can add Copper and Tracks for example along them.

Text Search in Find Bar

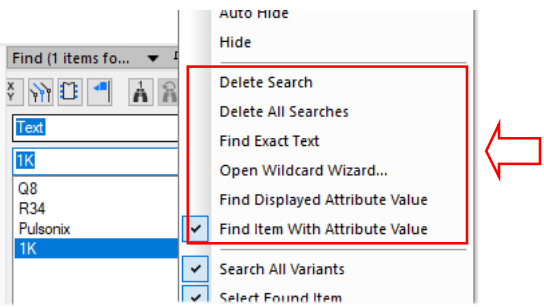
A **Text** search has been added to the **Find Bar,** this can find all 'free' Text and Attribute text.



Choose **Text** in the Find Bar as the '**type'** and enter the text to find in the box. As you search for different text items in your design, the searched list will be populated. This can be refined using the **Delete Search** option from the context menu on the selected text string.

Note, to find Component Names, like, R1, C1 etc. choose Attribute Text.

New options on the context menu are available for Find Text.



This option can also be used to search for attribute values by selecting the **Attribute Finder** from the context. You must select the Find Displayed Attribute Value if you wish to find an Attribute text.

Context menu commands

Delete Search - Deletes the selected search from the list.

Delete All Searches - Deletes all previous searches from the list.

Find Exact Text - Only text matching the exact search string will be found.

Open Wildcard Wizard... - Use this to add wildcards to the search string. Only enabled if **Find Exact Text** is disabled.

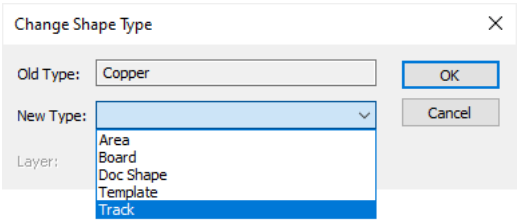
Find Displayed Attribute Value - Use this to find displayed attribute positions with a value matching the search string. This is automatically deselected if **Find Item Attribute Value** is selected.

Find Item Attribute Value - Use this to find items with attributes containing a value matching the search string. This is automatically deselected if **Find Displayed Attribute Value** is selected.

Change ‘open’ Copper to Track and vice versa

The ability to change an **‘open’ Copper Shape** to a **Track** has been added.

To use this option, select an ‘open’ Copper shape, right click and from the context menu, select **Change Shape Type**, choose **Track** from the drop-down list.

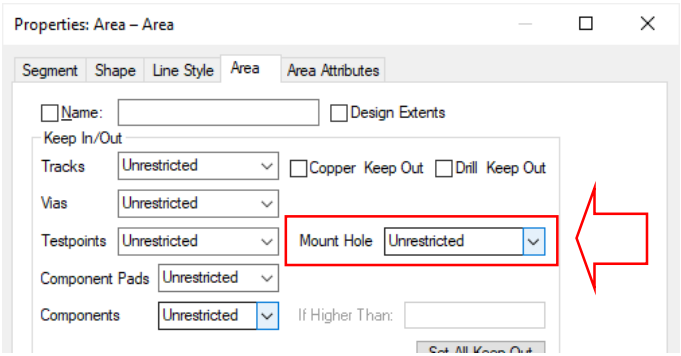


This can only be done if the selected shape is not a controlled shape e.g. spline.

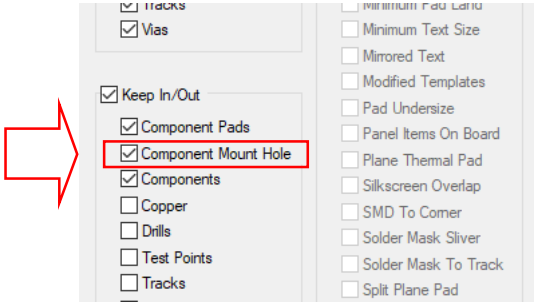
The reverse can be done to convert a **Track** to **Copper**.

Area Mounting Hole Keep Out/In

Properties of an **Area** has been modified to allow **Mounting Holes** as **keep In/Out.**

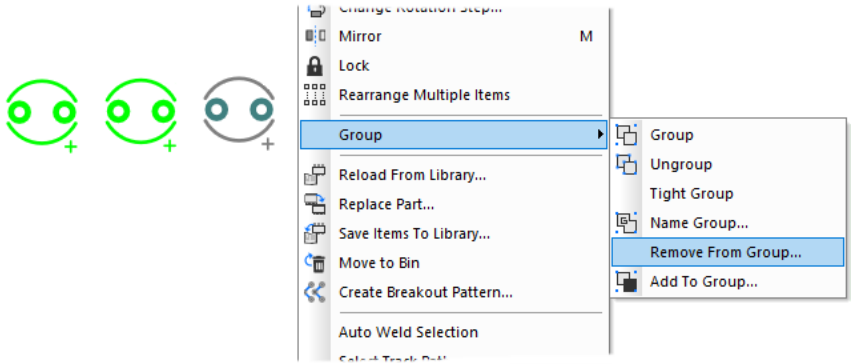


This new type will be included in the **Design Rules Check** for **Keep In/Out** rules under **Component Mount Hole.**



Remove from Group when multiple items are selected

The **Remove From Group** command is now available when multiple items are selected within a **non-Tight group**. It will only appear when components are all in the same group. If a component is part of multiple groups it will only be removed from the group common to all selected components.



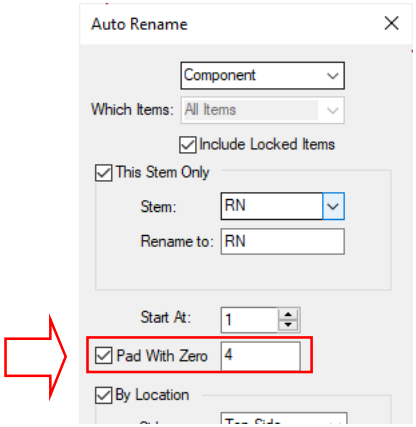
Match Rotation/Mirror of previous Component from Bin

When you drag a component from the Component Bin, its rotation will now match the last component of the same type dragged from the Bin. In previous releases, the rotation was not considered for any component.

Auto Rename - Padded with zeros

When using **Auto Rename**, the new name can be padded with zeros, so all the name lengths are the same. This can be achieved by checking the **Pad With Zero** box and setting the number of zeros for padding. For example, a resistor with **Pad with Zero** set to 4 would become R0001.

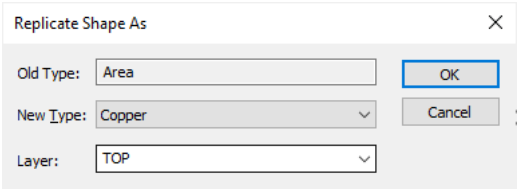
The only instances where this doesn't really apply is when the padding is set to a low number, such as 2, and the names could be more than 3 numbers, for example, C01, 02 to 99 and then C100 The lengths are now not the same.



New ‘Replicate Shape As’ feature

There is a new command, **Replicate Shape As**, that works similar to **Change Shape Type**, but instead, keeps the original shape.

To use this option, select a shape, choose the **Replicate Shape As** command from the context menu and then choose the **New Type** and **Layer**. The shape will be copied to the new layer and shape type, and will use the same shape position.



Star Point Design Settings Defaults Changed

Star Point Design Setting defaults has been updated to include a **Change Symbol** button and a **Name Stem**.

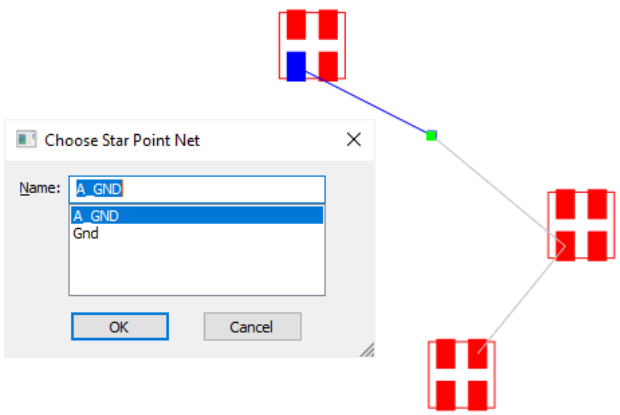
Highlight Net includes Star Point

In PCB designs, highlighting a net now also highlights **Star Points** on that net.

Find Bar options (Select, Highlight, Brighten, etc.) will also be applied to **Star Points** when searching for a net.

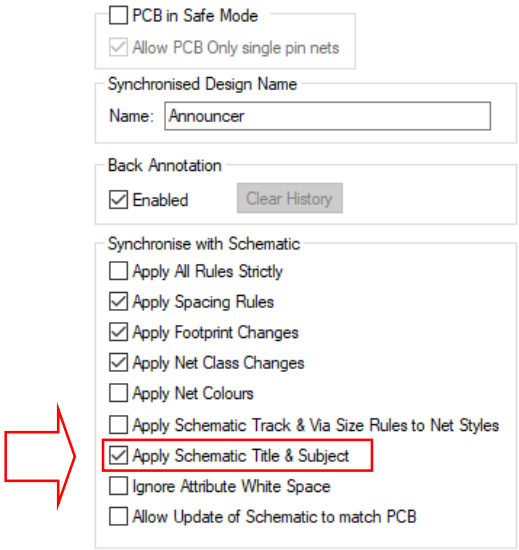
The **Mark Net** option now marks **Star Points** on the net.

Mark Net can be used when a Star Point is the selected item (in PCB). If the Star Point is on more than one net, the **Choose star point net** dialog will appear, allowing you to choose which net to mark.



Option to Synchronise the Document Title & Subject

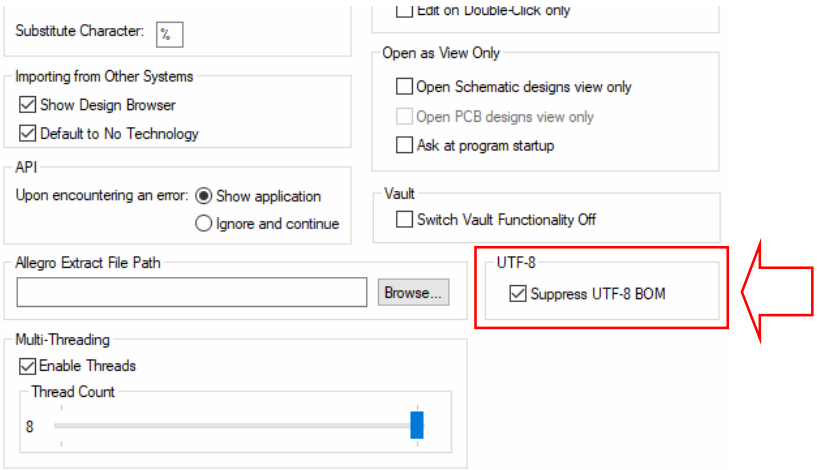
In the **Design Settings** and **Synchronisation**, you can select the **Apply Schematic Title & Subject**. This will allow the Title & Subject to be synchronised between your designs.



Option to Suppress UTF-8 BOM used in PLM and Scripting

A new check box has been added to the **Options** dialog and **General** page to **Suppress UTF-8 BOM**. Checking this option will prevent the byte order mark from being written to an output file. The check box is set as unchecked by default to prevent interference with current settings. If you wish to use it, you should check this option to enable it. This is required by customers using PLM software and Pulsonix scripting that writes a BOM which can cause an error when being read in. The file can be encoded with UTF-8 rather than UTF-8 with BOM.

This feature was back-fitted to 11

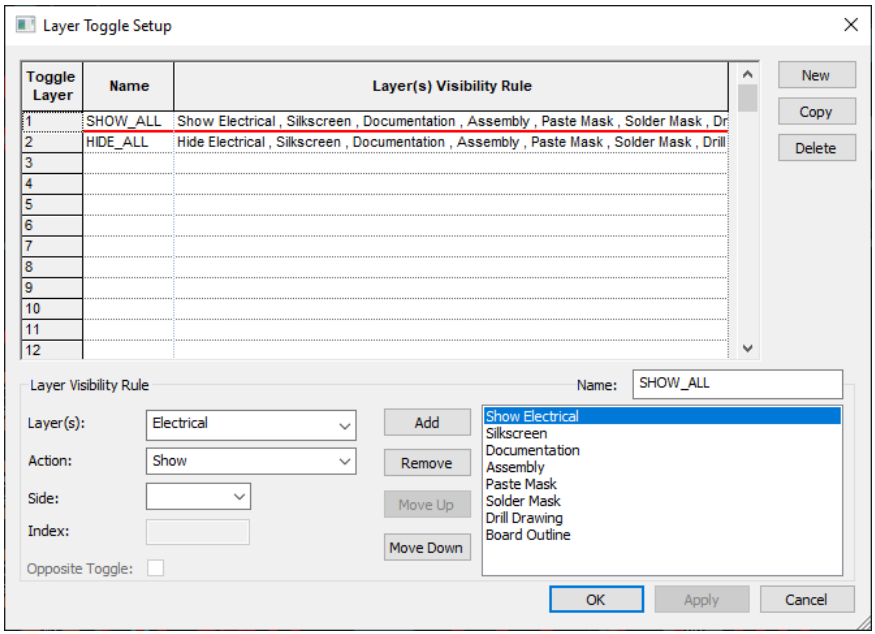


Toggle Layers

Redesigned Toggle Layer dialog

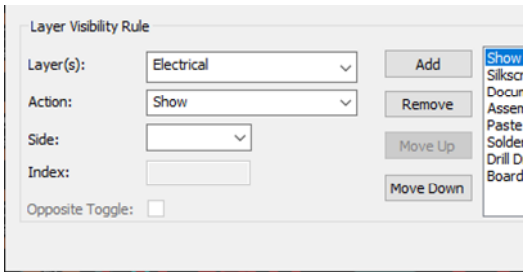
The **Toggle Layer** dialog has been revised to make it less programmer and more user-friendly.

The changes reflect the **Technology** dialog layout and use a grid to contain the different toggles, this includes the **Toggle Layer**, **Name** and **Rule**.



Three new buttons on the right side are used to manage the different toggles; create a **New** toggle, **Copy** and existing one and **Delete** a toggle layer.

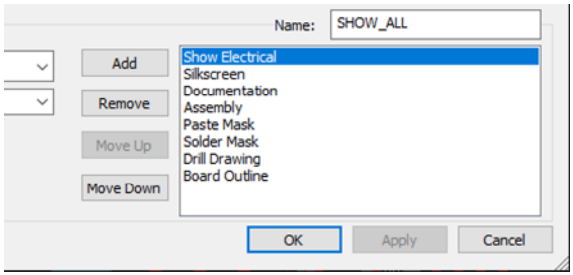
Below the grid there are various controls under **Layer Visibility Rule**, that help you to construct and/or edit a layer command with all controls shown (but enabled and disabled accordingly).



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

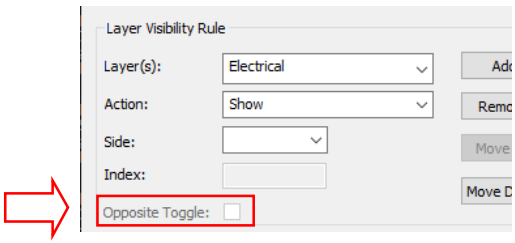
Omitting the Index when specifying **Inner** implies ALL inner layers, so "Electrical Inner" will toggle all the inner electrical layers.

There is also a list box that shows all of the layer commands within a toggle (each printed on a separate line) allowing you to easily select and edit layer commands by clicking on that layer command in the list. There are also 4 new buttons that help manage the list of layer commands.



Switching Multiple Layers On And Off Simultaneously

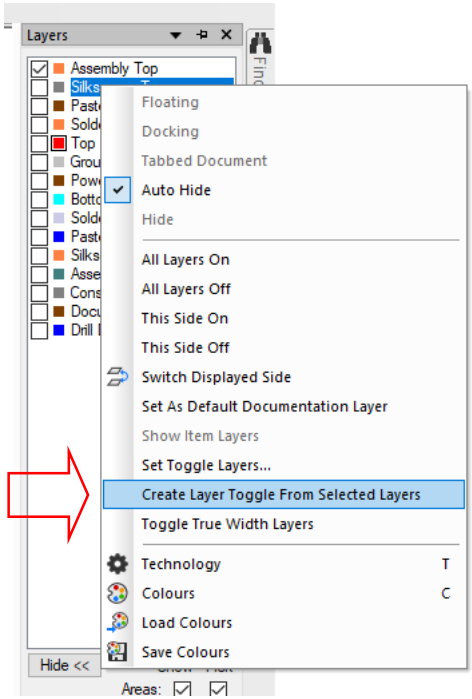
Use the check box **Opposite Toggle** to visibly switch a layer the opposite way to the first layer specified.



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

Layers Bar – Create Layer Toggle From Selected Layers

From the context menu on the **Layers Bar**, a new, **Create Layer Toggle From Selected Layers** option has been added.

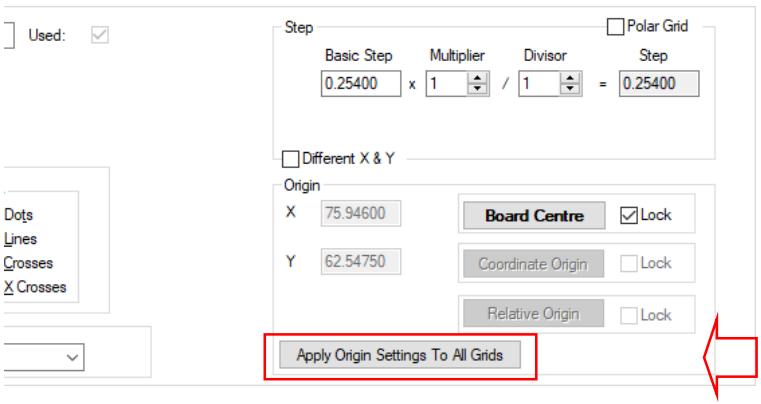


When selected, the currently selected layers in the bar will be added to a new **Layer Toggle** layer with the action **ShowEx**. This means that when this new toggle is clicked in the Layer Toggle Bar, only those selected layers will be shown, everything else will be hidden.

Technology Dialog Changes

Grids Page - Apply Origin Settings To All Grids

The **Apply Origin Settings To All Grids** check box is now a push button instead of a check box. The button is disabled if all grids have the same origin.

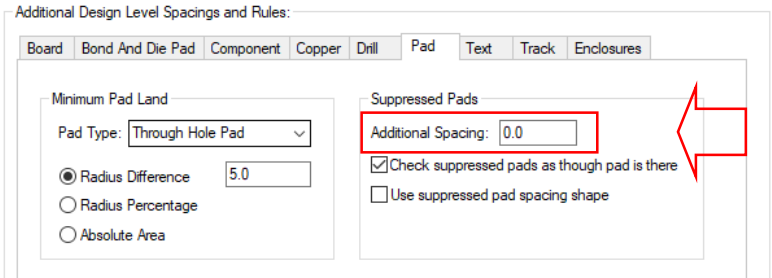


This feature was back-fitted to V11.0

Spacing rules - Design level - Pad - Suppressed pads - Additional spacing for Drill Holes

On the **Spacing Rules** under **Design Level** and **Pad**, an additional spacing around the **drill hole** of a **suppressed pad** can now be defined.

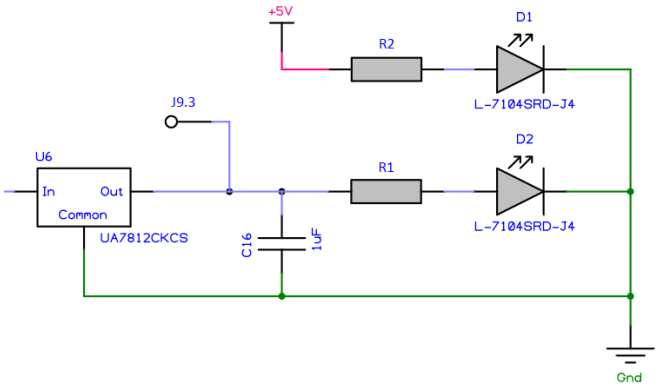
This is in addition to the spacing derived between the pad item (Pad, Mounting Hole or Via) and the other item. This is instead of the two options, **Check suppressed pads as though pad is there,** and **Use suppressed pad spacing shape.**



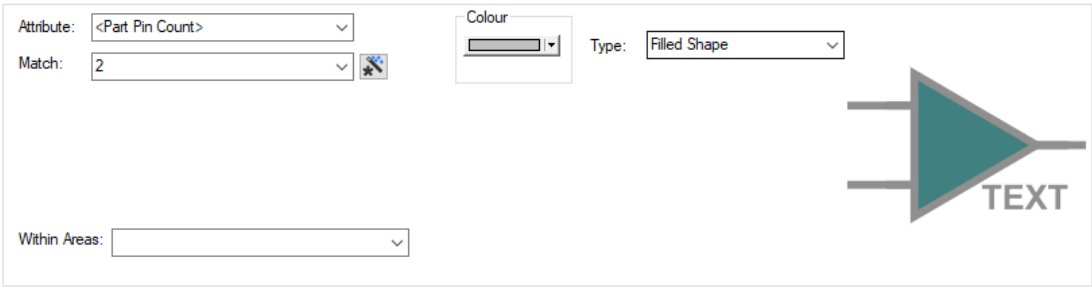
Component Colour Rules

A new tab in the **Colours** dialog has been added for **Component Rules**. This tab is essentially the same as a rules page except it is used to define colours for **Component Symbol Shapes** and/or **Text**.

This means it can be used in a Schematic for example, for colouring all two-pin devices as solid.



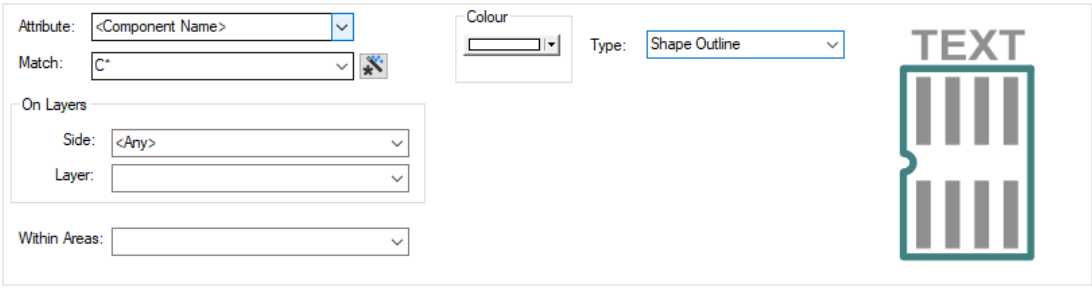
The **Schematic Colours** page for **Component Rules** allow you to **Match** parameters based on the **Attribute** as you do with any other rule.



The <Pin Count> attribute type is available to enable components so be matched to a rule given the number of pins.

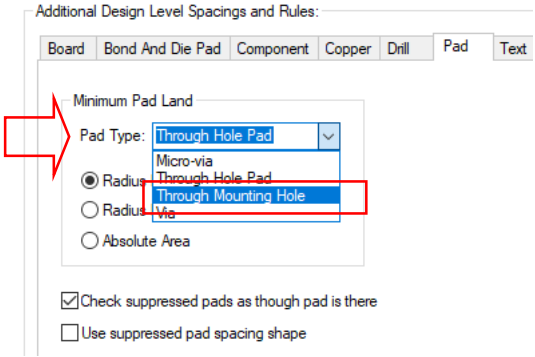
The PCB Colours **Component Rules** has slightly different combinations and **Layer** selections:

The new rules can be used in both PCBs and Schematics but have different options for each.



Minimum Pad Land for Mounting Holes

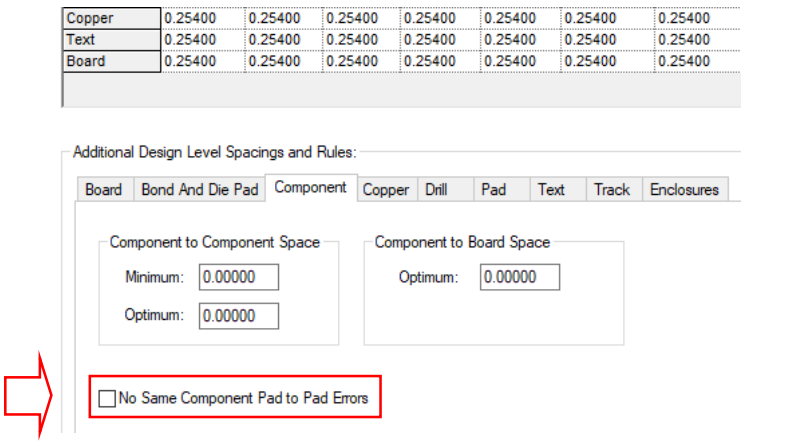
You can now define the Minimum Pad Land for Through Mounting Holes by selecting them on the Pad Type drop down list under **Technology**, **Spacing Rules**, **Design Level** and **Component** tab.



No Same Component Pad to Pad Errors extended to include Mounting Holes

The **No Same Component Pad to Pad Errors** option under **Technology**, **Spacing Rules**, **Design Level** and **Component** tab now includes **Mounting Holes**.

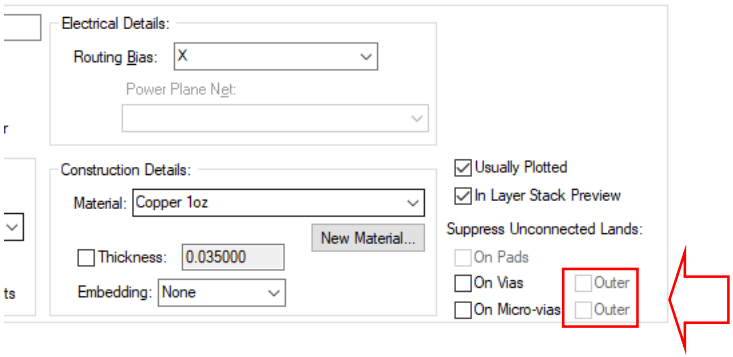
The **No Same Component Pad to Pad Errors** option check box is used to ignore pad to pad errors within the same component footprint (or documentation symbol), but now also includes mounting holes.



Suppress Pads on Outer Layers

You can choose to suppress unconnected lands on through hole **On pads**, **On Vias** or **On Micro-vias** on an electrical layer.

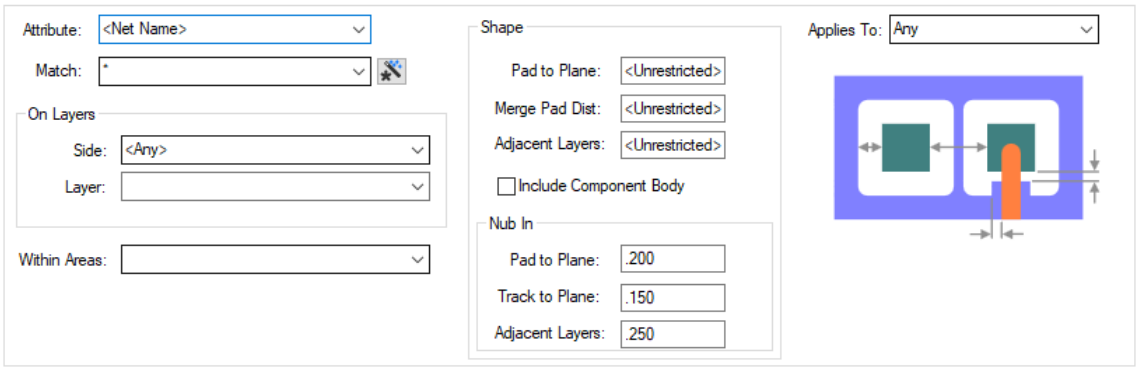
There is a new separate option to override the land suppression when on the **Outer** (top or bottom) layer of the span for **Vias** and **Micro-vias**. A land is unconnected if there is no track to it on that layer and it is not contained in a template of the same net, on that layer. The land is not drawn and will not be plotted.



Anti Pad Rules

The **Anti Pad Rules** dialog is used to specify the rules used to generate additional copper plane cutouts on layers below pads (or vias, mounting holes, etc). When copper is poured, or a powerplane generated, these cutouts are applied. This rules dialog allows the cutouts to be auto-generated.

There is a new page in **Technology** under **DFM/DFT** for **Anti Pad** Rules.



This is a specialist feature and won't be generally used by most users. It is used for high-speed designs with GHz signals. This is necessary to avoid parasitic capacitance. In some cases, the whole component area needs a cutout, the rules page allows this.

Below are some reference links regarding this topic:

http://www.ti.com/lit/an/snla074a/snla074a.pdf?ts=1591360735180

https://docs.broadcom.com/wcs-public/products/application-notes/application-note/980/533/av02-0725en-an_5362-27jul10.pdf

https://www.amphenol-aerospace.com/resources/literature/view/r-vpx-routing-guidelines

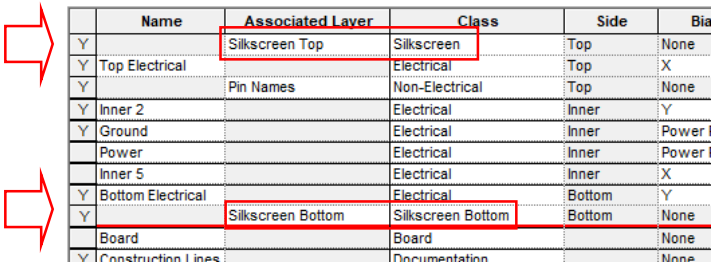
This feature is fully described in detail in the online help.

Associate a layer with a different non-electrical class to Swap With

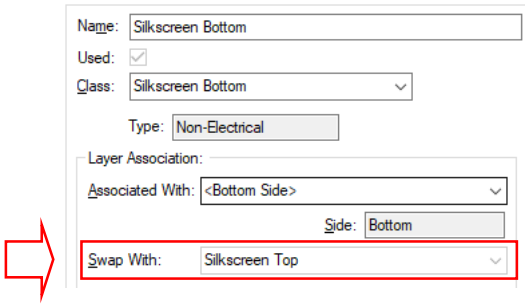
You can now associate a layer with another layer of a different **Layer Class**. Previously, this was not allowed and only matching Layer Classes could be swapped.

In the **Technology Layers** tab, it shows the layer to **Swap With**. If there is not an equivalent layer of the same class, you can associate a different Layer to swap with. This allows you to have different layer class settings on the 'other' side.

In the example below, **Silkscreen Top** and **Silkscreen Bottom** have different **Layer Class** names (each class containing different contents). The **Swap With** setting enables you to associate them without error.

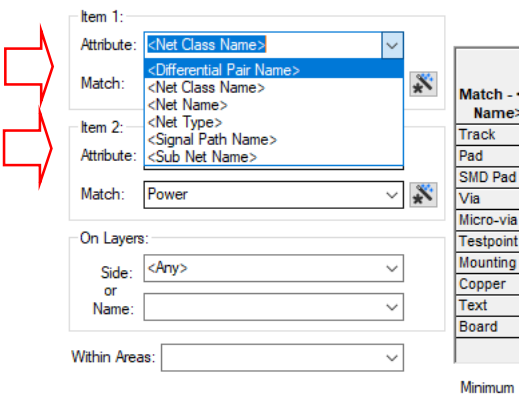


	Name	Associated Layer	Class	Side	Bia
Y		Silkscreen Top	Silkscreen	Top	None
Y	Top Electrical		Electrical	Top	X
Y		Pin Names	Non-Electrical	Top	None
Y	Inner 2		Electrical	Inner	Y
Y	Ground		Electrical	Inner	Power I
	Power		Electrical	Inner	Power I
	Inner 5		Electrical	Inner	X
Y	Bottom Electrical		Electrical	Bottom	Y
Y		Silkscreen Bottom	Silkscreen Bottom	Bottom	None
	Board		Board		None
Y	Construction Lines		Documentation		None



Match Pair Spacings for Diff Pairs, Signal Paths and Sub Nets

You can define **Match Pair Spacings** for the **Attributes <Differential Pair Name>, <Signal Path Name>** and **<Sub Net Name>,** as well as user defined Diff Pair, Signal Path and Sub Net attributes.



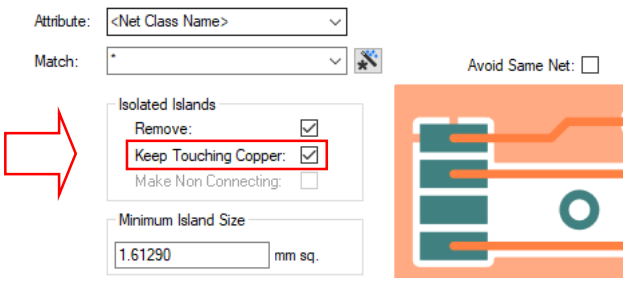
These defined spacings will produce valid errors when running batch **DRC**, and also when using **Online DRC** (with continuous Online DRC backing off from possible errors).

The **Rule** and **Spacing** controls on the **Measure** dialog **Gap** tab take into account these new Match Pair Spacings as well.

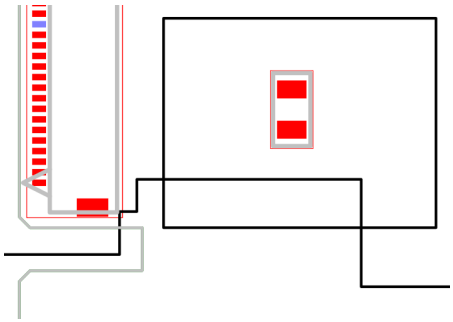
Copper Pour – Keep Touching Copper

Copper is normally considered not isolated if it touches another piece of copper on the same net. This changes the results in some cases, particularly when using nested templates on the same net.

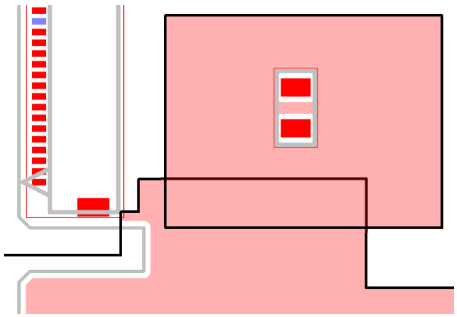
There is now an option in **Technology** on the **Copper Pour Rules** to make it Keep or remove the copper - **Keep Touching Copper**.



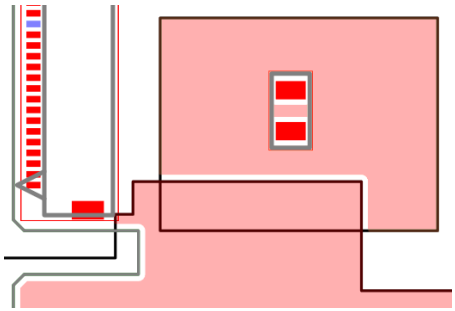
The images below show two overlapping templates, before pouring:



And after pouring with the **Keep Touching Copper** switch checked, he two shapes are poured touching (connected).



As a comparison, this is after pouring with the **Keep Touching Copper** switch unchecked:

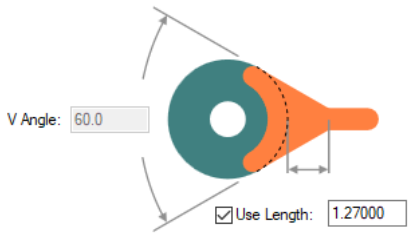


This feature was back-fitted to V11.0.

Define Teardrop by Length

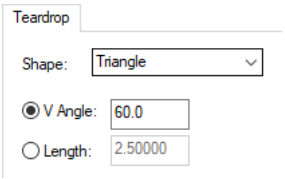
The teardrop size can now be defined by its **length** from the edge of the pad.

Where a teardrop is on a round pad for example, it can be too small when the V Angle is set to 90 degrees (with the angle determining the teardrop length), this feature helps eliminate this issue.



The length can be set in the **Teardrop Rules** in the **Technology**. This will be enabled by selecting the **Use Length** check box. The default length is 50 thou or metric equivalent. With Use Length selected, the **V Angle** setting is deactivated.

The length can be set on the **Properties** of **Teardrop** for a selected teardrop. This can also be used to toggle between defining size by **V Angle** and **Length** (as well as the **Shape** style).

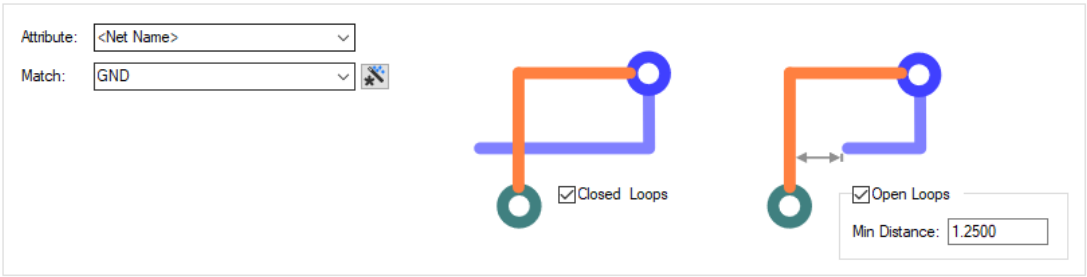


Loop Antennae Rule

A new rules page has been added to the **High Speed** section in the **Technology** for **Loop Antennae**. It will only be available if you have the Interactive High-Speed option and license.

This rule and subsequent DRC process, will check for closed and/or open track loops. You can also define a minimum distance for open loops which is the minimum spacing needed between the two tracks to not form an open loop.

A closed loop is formed when two tracks attached to a via on different layers overlap (i.e. they cross). An open loop is not a complete loop (closed) but still forms a loop area.



The appropriate **Loop Antennae** rule is found for each net item by working down the list of rules until the first match is found. Hence, the order of the rules is important. A match is when the specified **Attribute Name** and **Match Value** match attributes of a net item. The Loop Antennae rule must have the given attribute, and its value must wildcard match the value.

Closed Loops defines whether or not to check for closed loops.

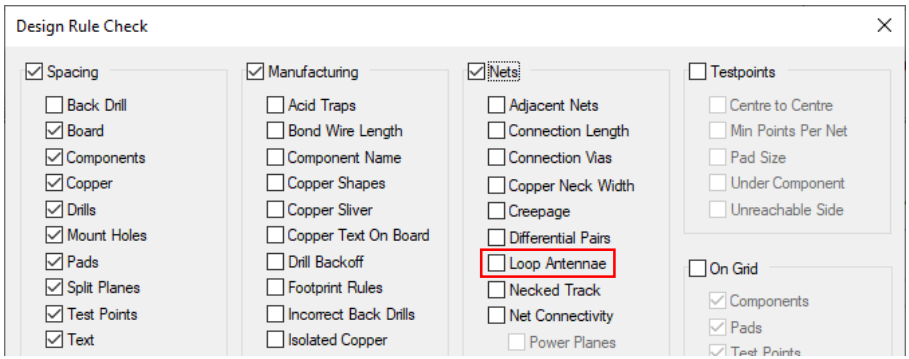
Open Loops defines whether or not to check for open loops.

Min Distance defines the distance required between a track across layers to not form an open loop. If the distance between any two segments across layers is less than this minimum distance and greater than zero, then an open loop is formed (a distance of zero would form a closed loop).

Design Rules Checking

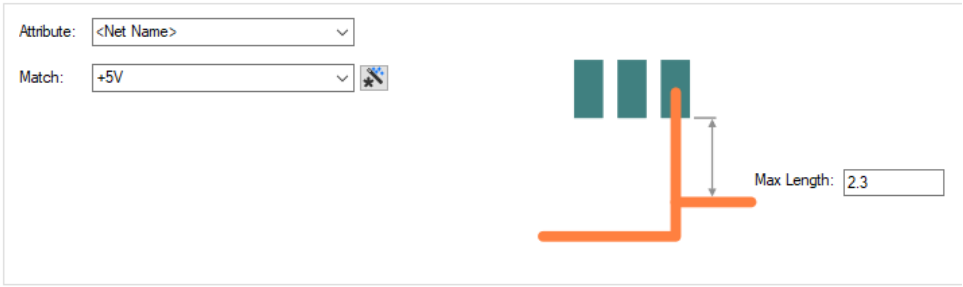
The **Loop Antennae** check has been added to the **Net** section in **DRC** (which produces the **LA** error code if in error).

This check finds closed and/or open loops (as defined by their rule) formed across layers without an intervening copper plane which covers the loop area.



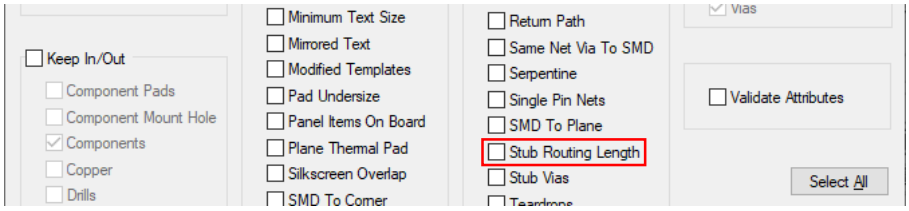
Stub Routing Length Rule

The **Stub Routing Length** rule has been added to the **DFM/DFT** rules in the **Technology**. From here, the maximum length is defined between the pad and the first track junction or via it comes to.



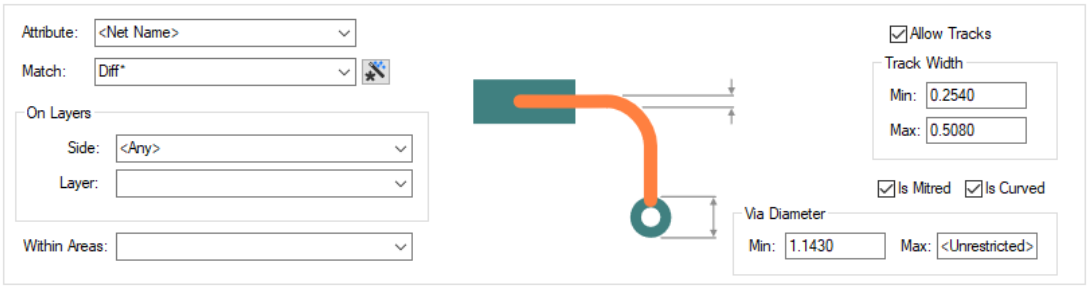
Design Rules Checking

The **Stub Routing Length** check has been added to the **Net** section in **DRC** (which produces the **SR** error code if in error). This check ensures the routed distance from the edge of a pad to another node (junction, pad, via etc) is not greater than the defined maximum value.



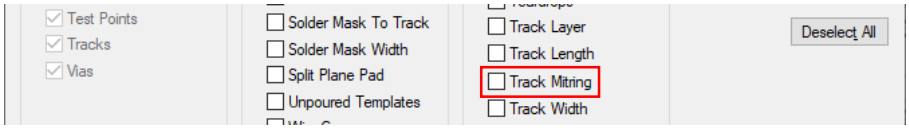
Track Mitring Rule

Two new rules for **Is Mitred** and **Is Curved** have been added to the **Track & Via Size Limit** page under **DFM/DFT Rules** in the Technology. This rule allows you to define whether all corners on rule-matching tracks are mitred, and can also ensure all mitres are curved.



Design Rules Checking

The **Track Mitring** check has been added to the **Net** section in **DRC** (which produces the **TM** error code if in error). This check ensures all corners on matching tracks are mitred, and can also ensure all mitres are curved.

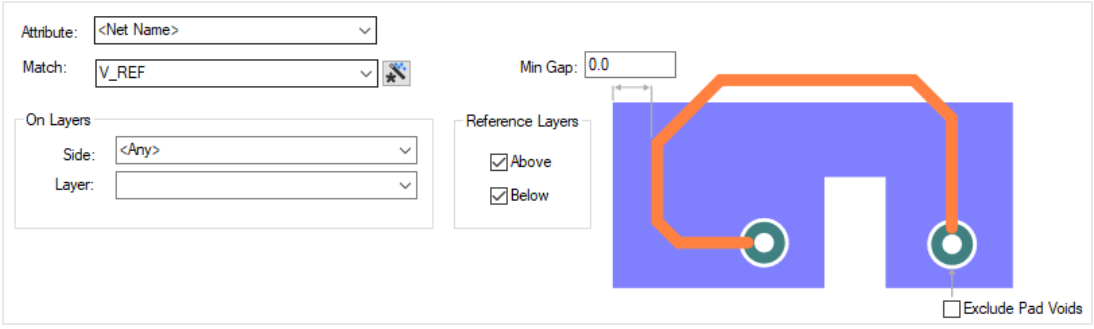


Return Path Rule

The **Return Path** rule has been added to the **High-Speed** rules section in the **Technology**. It will only be available if you have the Interactive High-Speed option and license.

This rule and DRC check ensures that track is covered by copper on defined **reference layers** (**Above** and/or **Below** the signal layer), and is greater than the defined minimum distance away from the edge of the copper (**Min Gap**).

You can also choose to **Exclude Pad Voids** which, if enabled, will not take into account Pad (or Via, Micro-Via or Mounting Hole) cutouts in the reference copper and it is on the same net as the track.

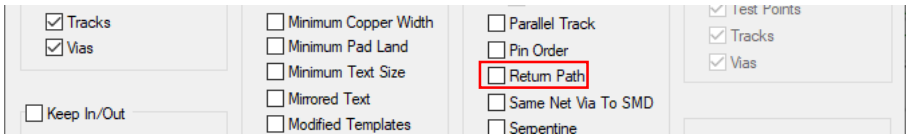


If there are any holes in the copper covering the track (could be made by a Pad cutout etc.) then a return path error will be found if the track is within the minimum distance of the hole. The **Exclude**

Pad Voids check box will not give an error if the item creating the hole is on the same net as the track (so a pad on the same net creating the hole).

Design Rules Checking

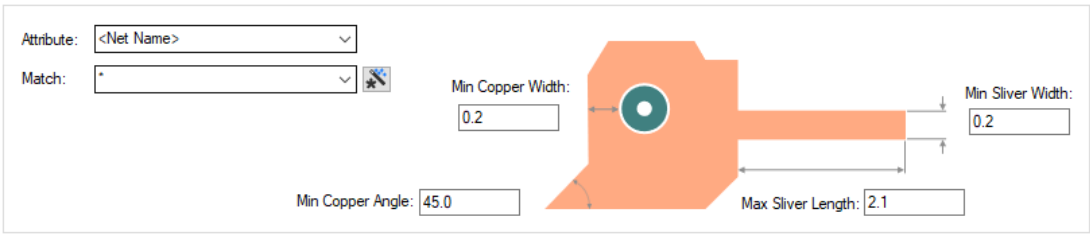
The **Return Path** check has been added to the **Net** section in **DRC** (which produces the **RP** error code if in error). The copper (and pad) are on an adjacent reference layer to the track (Above and/or Below the signal layer). The check is to ensure the track is completely covered by the copper, and you can define a minimum distance to stop the track from getting too close to the edge of copper.



Copper Check Rule

A new rules page has been added to the **DFM/DFT** section in the **Technology** for **Copper Check**.

Copper 'sliver' are as long, thin, dangling sections of copper that can cause issues during manufacturing. When creating high frequency designs, it can also act as a net antenna at certain frequencies.



This rule allows you to define parameters used to check for various sliver conditions. Although the width of a **Pour Template Area** will control most of these parameters, this rules page enables you to define and check them before plotting the design. Sometimes, slivers can inadvertently be created following pouring without knowing.

Min Copper Width is used to define the width of copper between two items, such as a pad cutout and another items' edge.

The **Min Copper Angle** is the minimum required angle between two copper segments to avoid a sliver being created. The angle between two joining segments has to be greater than this defined angle, otherwise an error will be found.

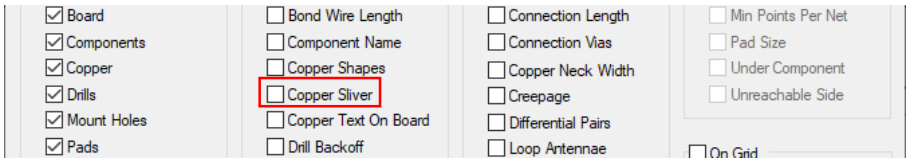
The **Min Sliver Width** is the smallest width possible for a section of copper to not be classed as an antenna. The **Max Sliver Length** is the longest the section of copper can be before it acts as an antenna.

A copper sliver will be a section of copper that is thinner than the minimum width and longer than the maximum length.

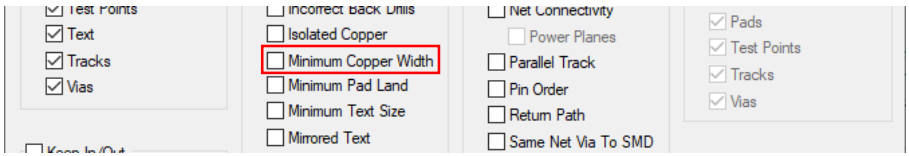
Design Rules Checking

The **Copper Sliver** check has been added to the **Manufacturing** section in **DRC** (which produces the **CoS** error code if in error).

This check returns an error if the distance between two copper cutouts (these cutouts can either be manually inserted cutouts, or cutouts around items when the copper is poured) is less than the defined value. For positive copper, the cutouts in the copper are also checked against the edges of the copper. For negative copper (powerplanes) all the openings around items are checked.



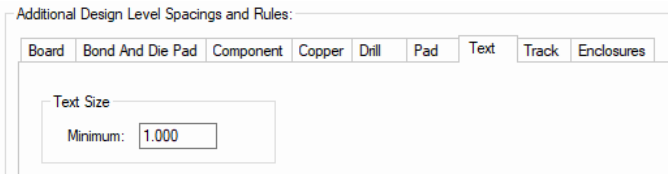
The **Minimum Copper Width** check will report an error code CoW



Minimum Text Size Rule

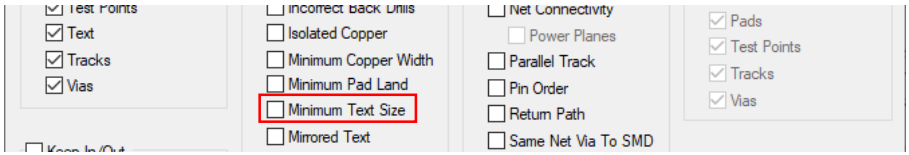
A **Text** tab has been added to the **Additional Design Level Spacings and Rules** section in **Technology Design Level Spacings**.

The minimum text size required for you design can be defined in the **Text Size** section. This prevents you from potentially creating a text size that is either unreadable or not manufacturable.



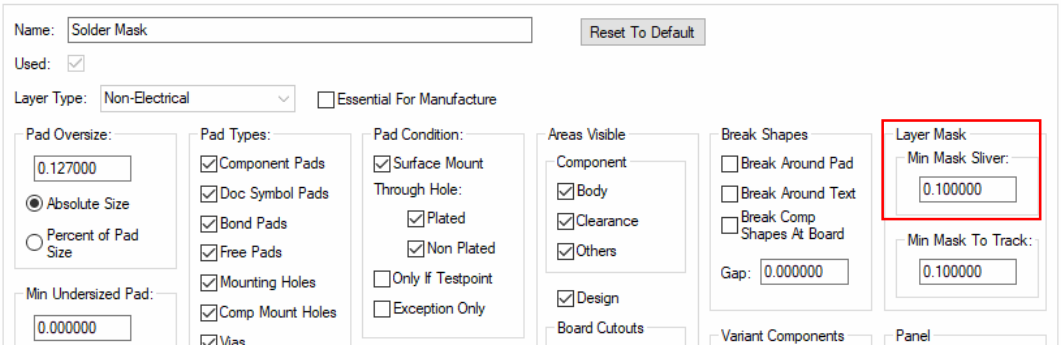
Design Rules Checking

The **Minimum Text Size** check has been added to the **Manufacturing** section in **DRC** (which produces the **TS** error code if in error). This check ensures all text in the design is larger than the defined minimum size.



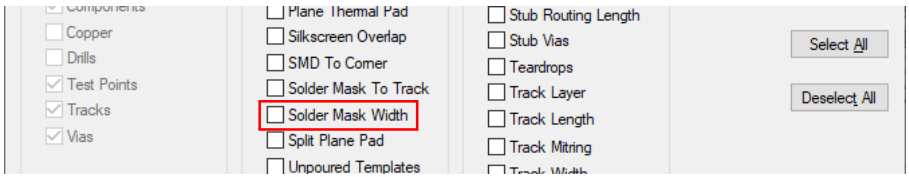
Min Solder Mask Width Rule

A **Min Mask Width** value has been added to the **Layer Class** dialog in the **Technology**. This defines the minimum solder mask width required. This value can only be defined on **Non-Electrical** layer classes.



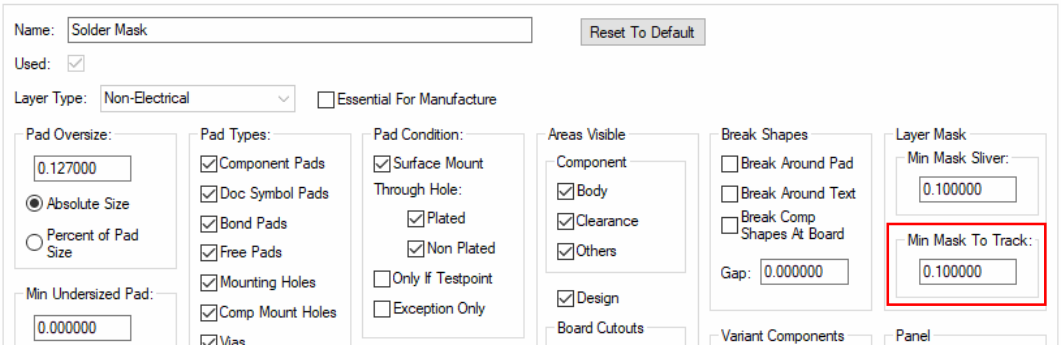
Design Rules Checking

The **Solder Mask Width** check has been added to the **Manufacturing** section in **DRC** (which produces the **SMS** error code if in error). This checks there is a minimum width of solder mask on any layer with a **Non-Electrical Layer Class** across the entire board that has the **Min Mask Sliver** defined.



Solder Mask To Track Rule

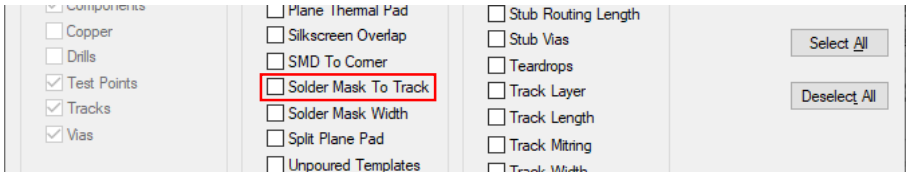
A **Min Mask to Track** value has been added to the **Layer Class** dialog in the **Technology**. This defines the minimum solder mask to track width required.



The solder mask to track check will check any layer with a layer class that has the Min Mask To Track value defined (this value can only be defined on **Non-Electrical** layer classes).

Design Rules Checking

The **Solder Mask To Track** check has been added to the **Manufacturing** section in **DRC** (which produces the **SoT** error code if in error). This check ensures there is a minimum distance between a track and a solder mask opening (not including the opening for the tracks start and end nodes if they exist).

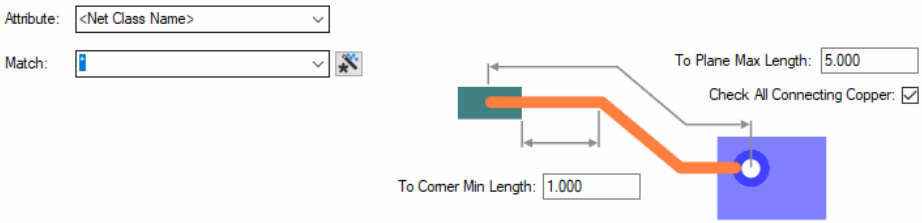


SMD To Corner and SMD To Plane Rules

The **SMD** rule has been added to the **High-Speed** rules section in the **Technology**. It will only be available if you have the Interactive High-Speed option and license. Within this, two new rules have been defined:

SMD To Corner check

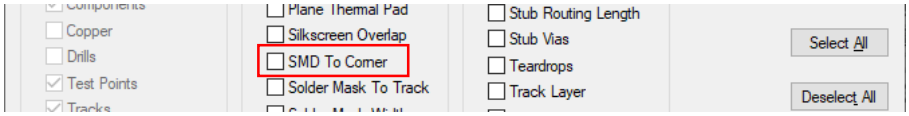
Using the **SMD To Corner** rule, you can define a minimum length of routed track measured from the edge of the SMD pad to the first corner of the track.



Design Rules Checking

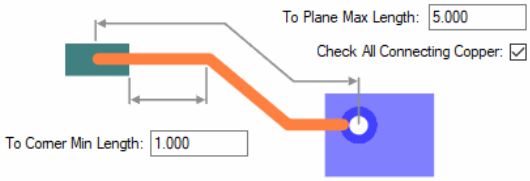
This check is used by the **Online DRC** when adding Tracks to your design and when running the batch **DRC** option.

The **Solder To Corner** check has been added to the **Manufacturing** section in **DRC** (which produces the **SM-Cnr** error code between the SMD pad (edge) and the corner if in error).



SMD To Plane check

This new rule allows you to define a maximum routing length from an SMD pad to a Via (or Micro-Via or pad) that is connected to a power plane.



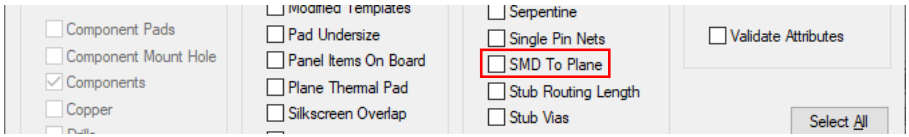
This distance takes into account the thickness of the board between the current layer and the power plane layer. The rule can be defined for a Net or a Pad style.

Check All Connecting Copper

Checking this option causes layers with a **Routing Bias** of **Power Plane**, and **poured copper** to be taken into account when checking **To Plane Max Length**. Unchecking this option will cause only layers with a Routing Bias of Power Plane to be checked.

Design Rules Checking

The **SMD To Plane** check has been added to the **Nets** section in **DRC** (which produces the **SM-Pl** error code between the SMD pad and the plane if in error).



New System Attributes Available

Added Last Saved Author Attribute

The system Attribute <**Last Saved Author**> can now be added. This can be used for showing the author on a drawing blank for example.

Added Last Saved Year attribute

The system Attribute <**Last Saved Year**> can now be added. This can be used for the copyright string for example.

New in-build attribute of <Not Fitted In> added

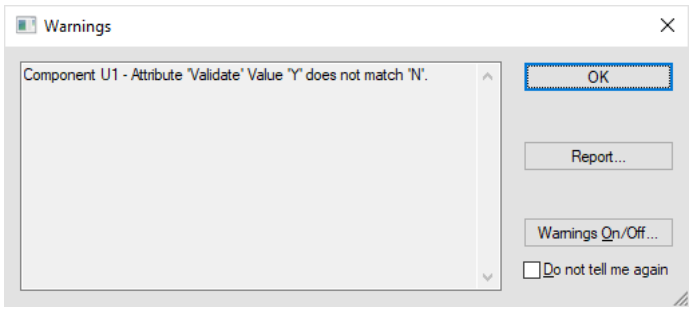
A new system attribute of **<Not Fitted In>** has been added for use with **Variants**. This is the opposite of the existing system attribute <Fitted In>.

When creating variants, it is sometimes advantageous to display the Variant that a component is not fitted in to highlight its status.

Attribute Validation when adding a Component

Using the **Attribute Validation** switch, attributes are now validated when a **Component** is added to the design.

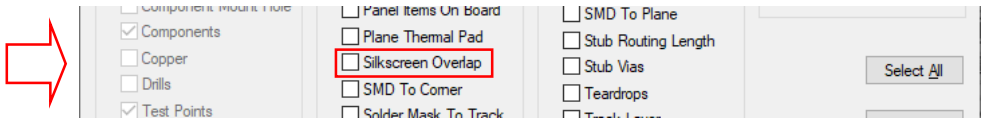
When used, it issues a warning message which can be turned off if required.



Additional DRC Checks

Silkscreen Overlap Doc Shapes

The **Silkscreen Overlap** check under the **Manufacturing** checks in **DRC** was available in V11.0 but it didn't take into account Doc shapes on a Silkscreen layer with overlapping pads. It does now and it will display an error when DRC is run. Before the Silkscreen Overlap check, it would only check for text on a Silkscreen layer so this expands the DRC for these items.



This feature was back-fitted to V11.0

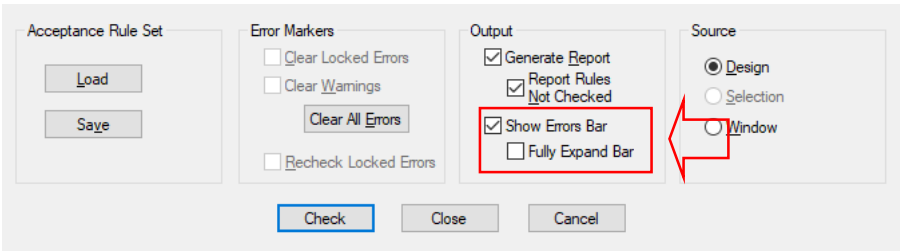
Check for Modified Templates

A new check for **Modified Templates** has been added to the **Manufacturing** section in DRC.

When run, this check will give an **MT** error if a template has been modified (i.e. the template shape has changed, items inside template have changed or moved etc.) but the template has not been repoured. This check should be used when **Repour Affected Templates** is disabled.

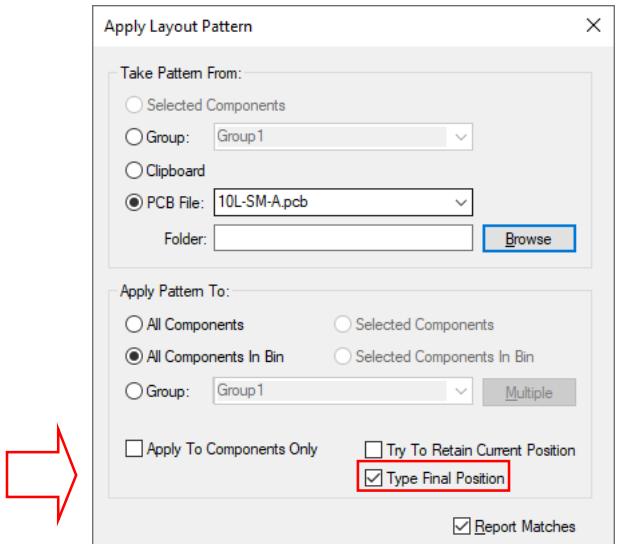
DRC Dialog – Shows Errors Bar can now show Fully Expanded Bar

When running the DRC option, there is an additional switch under Show Errors Bar, if when checked, enables you to Fully Expand Bar. This enables you to see all the error markers for all the layers and categories without requiring to expand each one.



Apply Layout Pattern – Type Final Position

A new option in Apply Layout Pattern called **Type Final Position,** enables you to type in the coordinates of the final position of the pattern.



You will be prompted once the layout process has finished, you can still place manually if you cancel out of the dialog. This is useful if you know where you want to place the pattern, and may be quicker that placing it manually.

Editing Differential Paired Tracks (High-Speed option)

Editing the **end of a paired track** to add some more segments has been enhanced to work as follows:

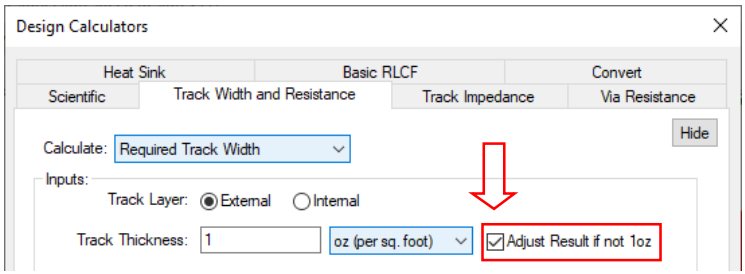
In **Select Mode**, double-click on paired track end always edits the paired tracks (it used to depend on if you picked within its width), and double-click on the connection attached to paired tracks will now also do the same. **Ctrl-double-click** is new and will always enter the start new pairing mode (or mirror mode depending on what option you have set).

In **Insert Track Mode**... this works the same as above. Click on connection or end of paired track where it will edit the end of the paired tracks. Ctrl-click will enter the start new pairing mode (or mirror mode depending on what option you have set).

This feature was back-fitted to V11.0

Design Calculators - Track Width Calculator Changes

Within the Design Calculators, a new check box has been added to the **Track Width and Resistance** dialog. To the right of the Track Thickness controls, a check box named **Adjust Result if not 1oz** is now available. If checked, it applies a correction to the formula for when the copper thickness is not 1oz copper.



Reasoning

If 1oz copper thickness is plugged into the current formula, the results exactly match the IPC-2221 results. The basic principle is, a track that is 2oz thick has half the resistance of an equal 1oz thick track, but almost exactly the same surface area to the board and the surrounding air.

If between 1oz and 2oz the resistance is halved then it is able to supply sqrt(2)=1.41 more current to produce the same heat ($P = I^2 * R$), but the surface stays almost the same, so the heat transfer should be almost unchanged between 1oz and 2oz.

The current capability increase for equal temperature rise should therefore be very close sqrt(copper-thickness / 1oz) . However, the IPC-2221 formula doesn't account for this as it only appears to only be intended for 1oz thick copper.

Solution

To account for this, the calculation solution is to always use 0,035mm (1oz) for the cross-section parameter, that the IPC221-formula requires, and afterwards the current is adjusted to account for the different thickness by multiplication with the factor sqrt(copper-thickness / 1oz), or in metric units: sqrt(t / 0,035mm).

Default Setting

This setting is now set to **On** by default but can be set off if you wish to revert back to the previous calculation.

Reference

Mr. Haberzettl of FuG in Germany has very kindly provided the adjustment formula and has run physical temperature tests to prove the correct results. With the corrected formula, the calculated temperature rise comes much closer to his measured result.

This link was provided from him with a more detailed explanation:

https://www.adam-research.de/pdfs/TRM_WhitePaper2_TraceTemperature.pdf

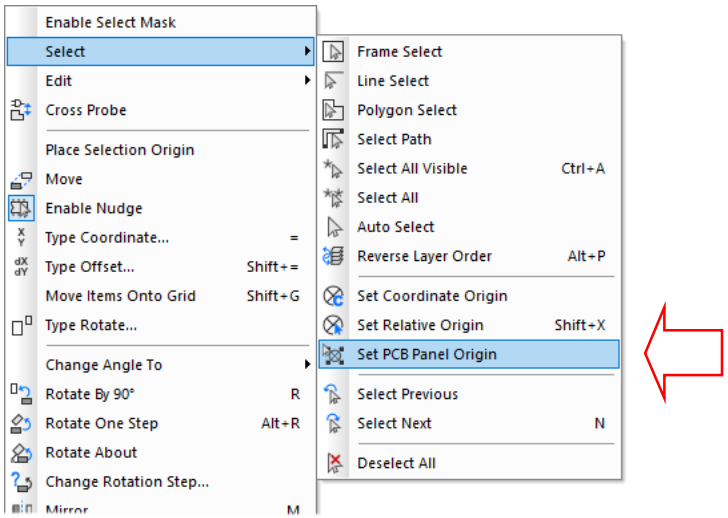
This feature was back-fitted to V11.0

Panel Editor Changes

Set PCB Panel Origin Command

In a PCB design, a **Set PCB Panel Origin** command has been added to the context menu. This has been added to enable you to set the PCB panel origin to a preselected item position instead of having to place it manually.

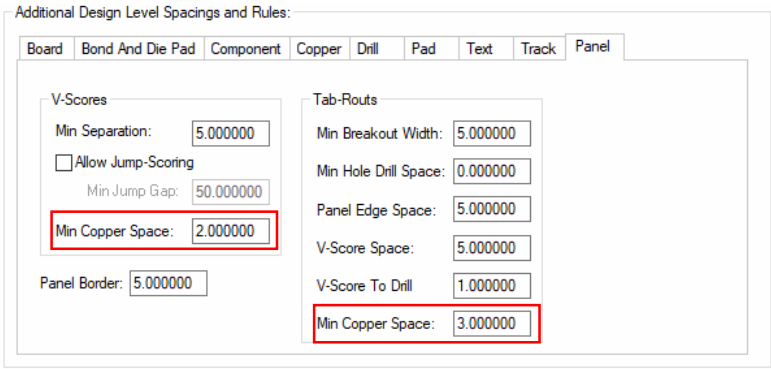
If you have a point selected in your design, a Pad, Mounting Hole, Construction Line junction for example, you can use the **Set PCB Panel Origin** command from the context menu. This command can also be used with nothing selected in the design, right clicking and selecting Select, then this option. A point would then need to be selected for the reference.



When adding PCBs to a panel design it means this Panel Origin can be use to position the board to an existing reference point, such as a Construction Line.

Min Copper Spacing for Tab Routs and V-Scores

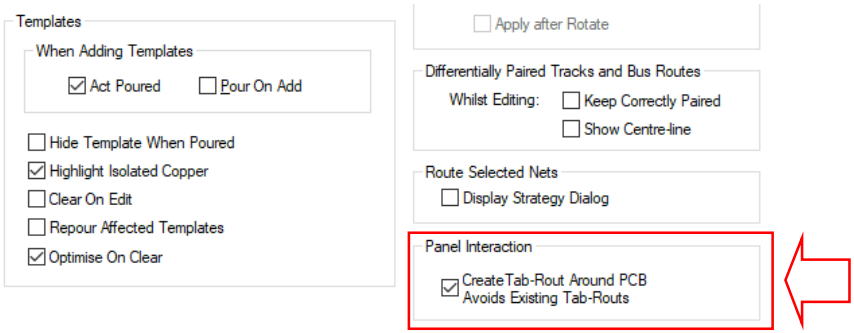
Within the **Panel Editor**, with the **Technology**, **Spacings** dialog and under **Additional Design Level Spacing Rules**, on the **Panel** tab a new spacing **Min Copper Space** has been added to both the **Tab Routs** group and **V-Score** group.



This allows the definition of spacing between Tab routs and Copper, and V-Scores and Copper. The spacing will produce errors when running the **Copper** spacing check in DRC. These new spacings are taken into account when pouring copper (in the Panel Editor).

New option to Create Tab-Route Around PCB Avoids Existing Tab-Routs

There is a new option on the **Options** dialog and **Interaction** page for a **Panel** design. **Create Tab-Route Around PCB Avoids Existing Tab-Routs**.



By default, this option is checked (to Create Tab-Route Around PCB) so that it backed off from existing tab-routs by the min breakout width (from Design Spacings). Unchecking this option will allow the tab-routs to flow around the board with no gaps to the existing tab-routes.

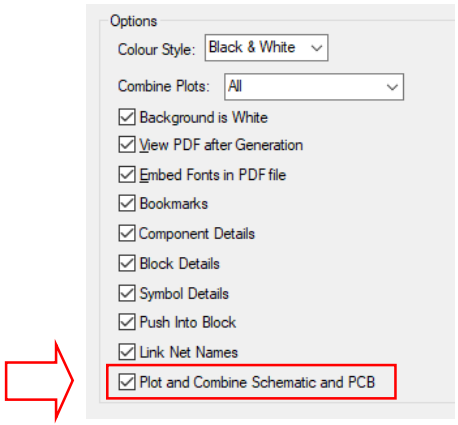
This feature was back-fitted to V11.0

CAM Plot Changes

PDF Export – Plot and Combine PCB and Schematic designs

You can now combine a Schematic and PCB plot in the same PDF plot file.

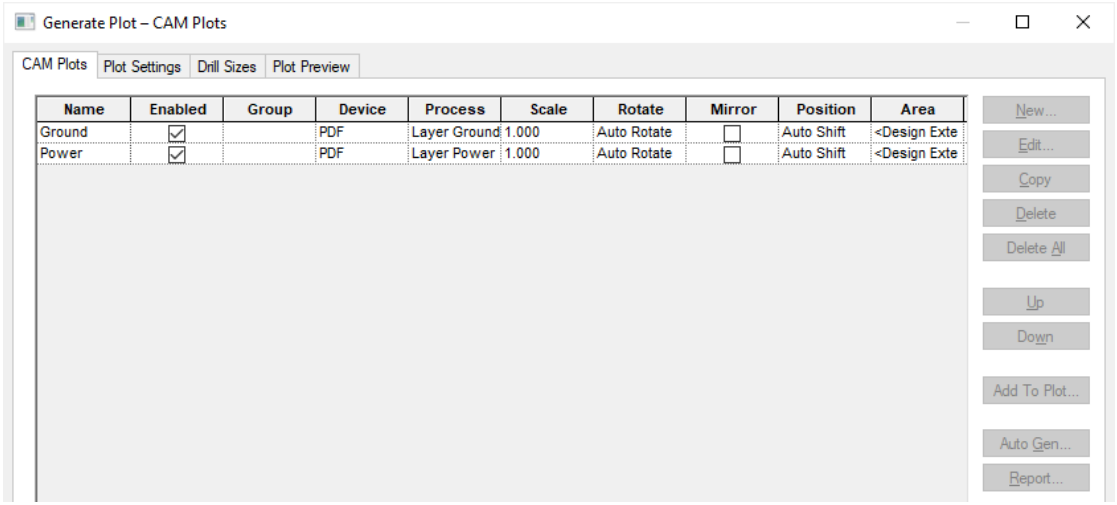
To do this, you must first enable the **Plot and Combine Schematic and PCB** design option in the **PDF Setup** option in **CAM Plots**. This setting is shared between both SCM and PCB applications, so can be selected in either.



A new **Plot and Combine Schematic and PCB** design option has been added to the **PDF Setup** menu in **CAM Plots**.

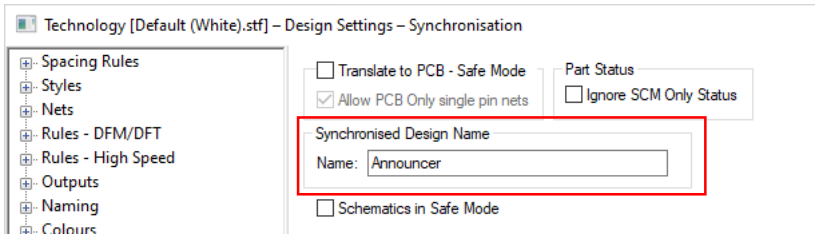
With this new switch enabled. when **run** is pressed, the current design plots are plotted and a new **CAM Plots** dialog opens allowing you to select what PDF plots you would like to plot from your linked design.

The linked CAM Plots dialog is partially disabled to only allow you to select or deselect the plots required. It is presumed that the plots would have already been defined in the design and at this stage just require plotting, not defining. If the plot appears incorrect, then cancel the plotting option and redefine the plots in other design.



Once **run** is pressed again, the dialog closes and the selected plots will be plotted and then appended to the end of the PDF document created. You can do this either way around; SCM to PCB, or PCB to SCM.

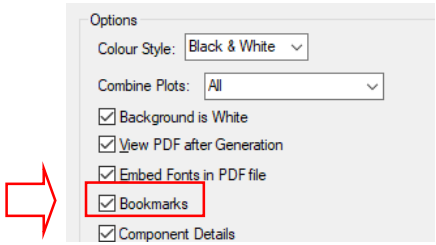
When running a combined plot, the program will either use the same name as the 'other' plot or will use the design name defined in the **Design Settings**, **Synchronise**, **Synchronise Design Name** option.



CAM Plot reports from both PDF plotting will be combined into one report.

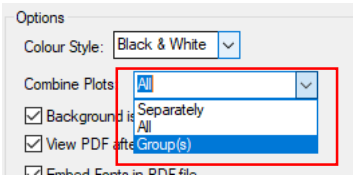
PDF Export - Plot and Combine PCB and Schematic designs share Bookmarks

When using the Combine SCM and PCB plot feature, Bookmarks created will be also combined provided the **PDF Setup** option, **Bookmarks** option is enabled.

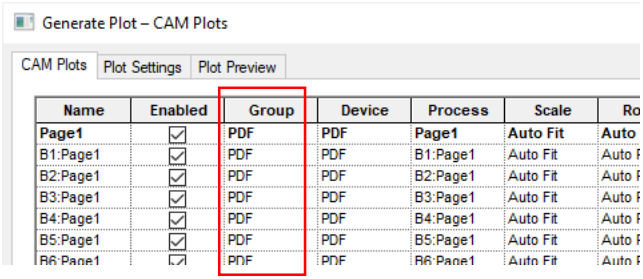


PDF Export - Combine plots by Group

Within the **CAM Plots** dialog and **PDF Setup** page, the **Combine Plots into Single Document** switch has been replaced with a new **Combine PDF** option containing a drop-down list.

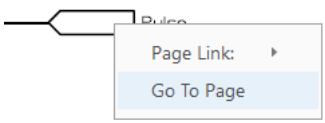


As with the old option, selections are still available for Plot **All** and **Separately** but this list now also includes Group(s). This command enables you to combine plots by the named plot **Group** selection for the plot.



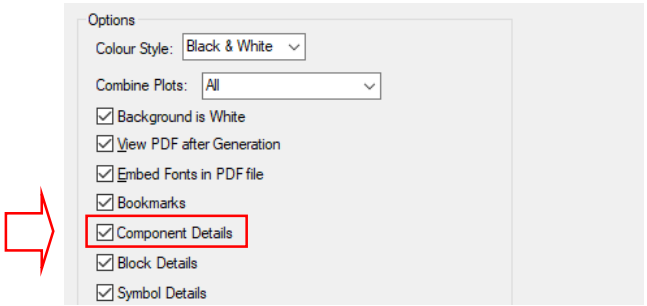
PDF Export - Allow click on page link symbol to go to page

When creating a **PDF Plot** using **CAM Plots**, if your schematic design has page link symbol(s), then clicking on the symbol in the **PDF document** will give you the option to **Go To Page** which will take you to the linked page.



PDF Export - Selectable Hyperlinks

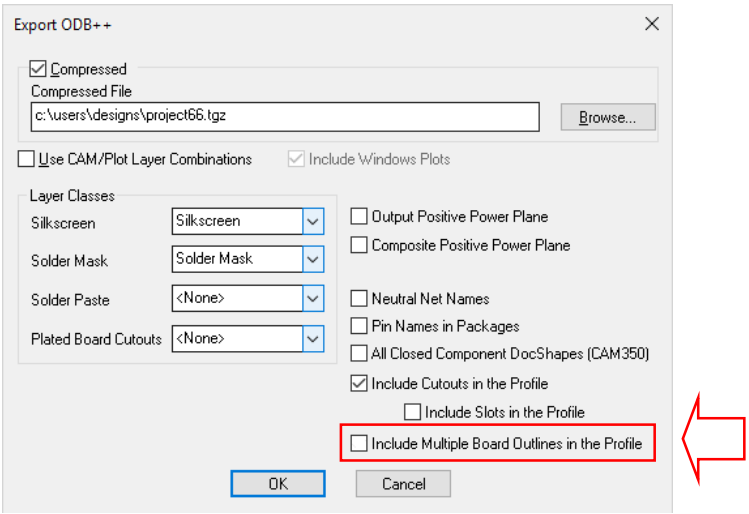
In an **intelligent PDF** document, for components showing their details, such as attribute values. If the component has a **<Hyperlink> attribute**, you can click on this attribute and it will open the link (provided it is valid).



This is a feature which needs the option **Component Details** in the **PDF Settings** page enabled and have a component with a <Hyperlink> attribute.

ODB++ Export Changes

There is a new switch on the **ODB++ Export** dialog, **Include Multiple Board Outlines in the Profile**.



In the ODB++ format, the board outline is defined and drawn as a 'Profile'. Most ODB++ viewers require the profile to be a single shape. Some ODB++ viewers have the ability to read multiple boards outlines in Profiles.

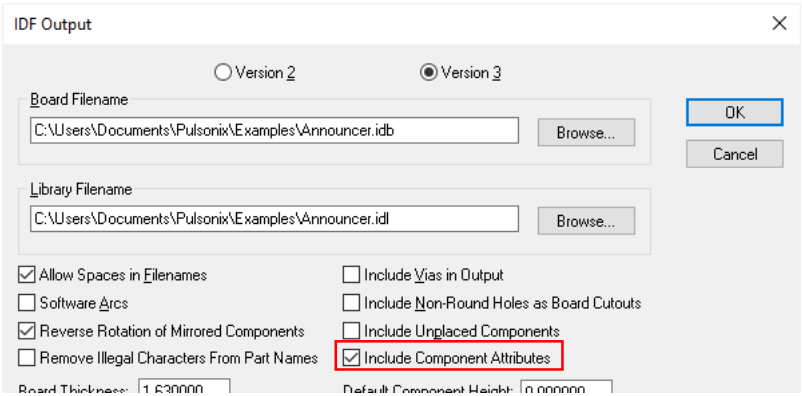
Check this switch to enable the output of all board outlines in a PCB design to the Profile, if unchecked only one of the board outlines will be included.

This feature was back-fitted to V11.0

Changes to IDF Output option

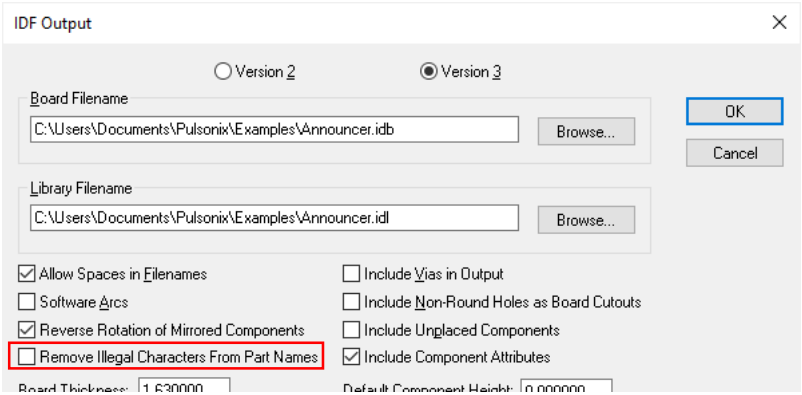
Option to not include Component Attributes

There is now a check box to enable/disable including Part attributes in the IDF output file. This option is enabled by default. This is only available for version 3 IDF files as Part Attributes are not included in version IDF 2.



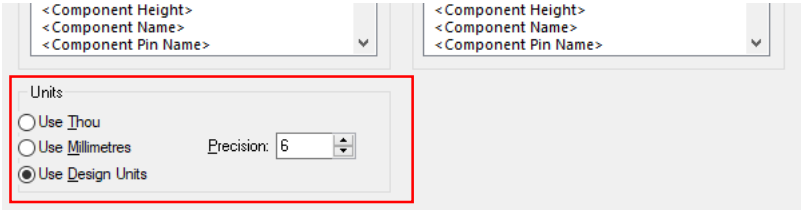
Remove Illegal Characters from Part Names

A new check box can be used to remove characters that are illegal in Part names for some programs, such as Inventor 3D. They will be replaced with an underscore. These characters include: \, /, :, *, ?, ", >, <, |



Change Units and Precision

There is now a **Units** option in the **IDF** output dialog. This means that where **Units/Precision** levels different from your defaults can be used in the IDF output. The default design units can still be used by selecting the **Use Design Units** radio button. Thou and Millimetres are the only unit options available and match the IDF spec.



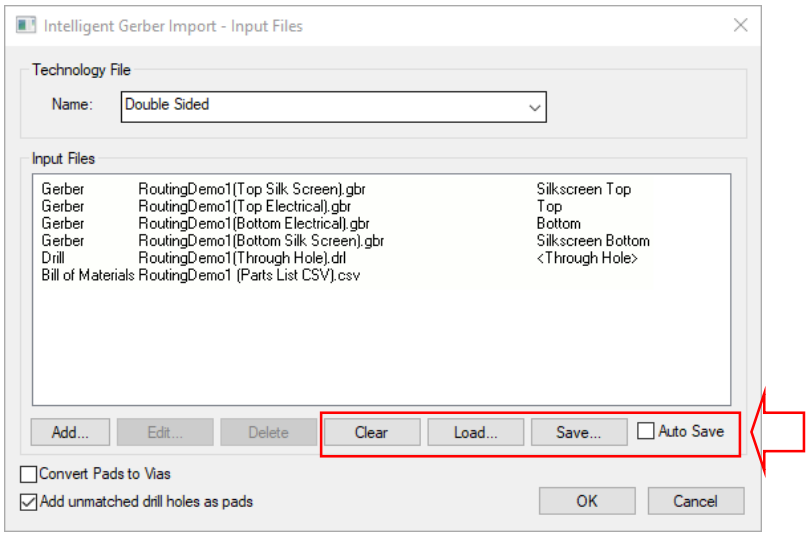
Changes to Intelligent Gerber Import option

Save/Load Settings

The **Intelligent Gerber Import** option now has ability to **Load/Save** and **Auto Save** dialog lists and settings.

Your current list and the settings within each dialog can be **Saved** and **Loaded** for subsequent use. The new Clear button will clear the whole list.

The new **Auto Save** button will save the current list. When you reopen the dialog, those settings will be recalled.



Import NC Drill file Only

The **Intelligent Gerber Import** option now now has the ability to **read** in just an **Excellon NC Drill file**. This enables the drill positions to be verified.

Import BOM

The **BOM file Import** now recognises multi ref name format, such as D1 D2 D3 etc when defined in quotes between comma fields.

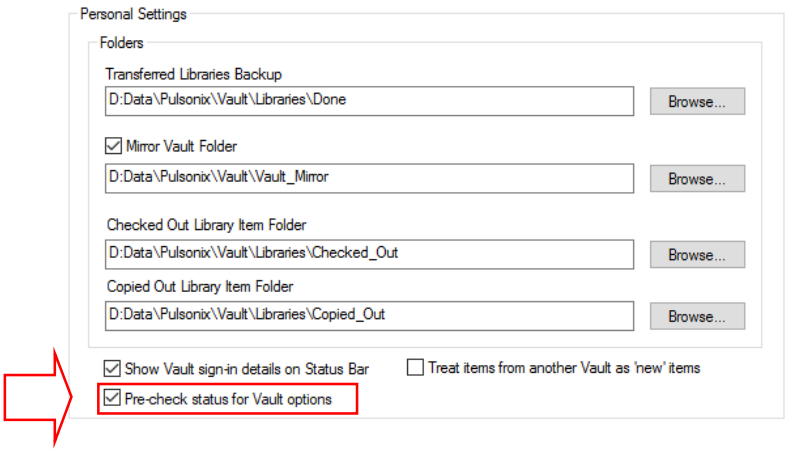
The **BOM file Import** now also recognises the Pulsonix naming convention C[1-9] and will resolve this to 9x individual component names, C1, C2….C9 etc.

Gerber files and G90 code

The G90 command (absolute mode) is now recognised when used in Gerber files.

Vault Setup – Pre-check status for Vault Options

New **Pre-check status for Vault options** setting for use when a slow connection, such as a VPN, is being used to access the Vault. Unchecking this setting stops a menu or toolbar from checking with the Vault to determine whether or not a status specific option, such as **Uncheck from Vault,** should be enabled. This avoids any undesirable delay in the display of the menu or toolbar.

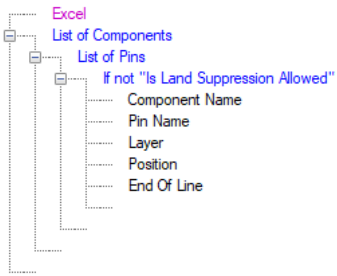


Report Maker Changes

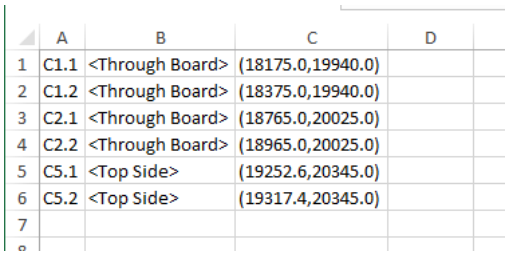
New command - Output to Excel

A new command has been added to allow the creation of Excel based reports.

The new **Excel Command** defines the report to be written in and saved as a .xlsx file.



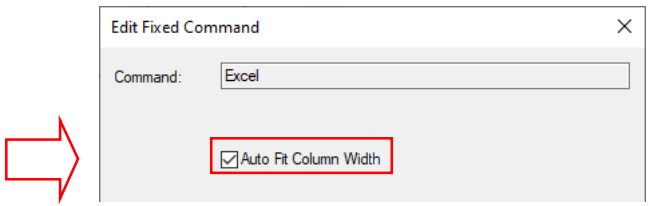
This will export the report directly and open Excel:



	A	B	C	D
1	C1.1	<Through Board>	(18175.0,19940.0)	
2	C1.2	<Through Board>	(18375.0,19940.0)	
3	C2.1	<Through Board>	(18765.0,20025.0)	
4	C2.2	<Through Board>	(18965.0,20025.0)	
5	C5.1	<Top Side>	(19252.6,20345.0)	
6	C5.2	<Top Side>	(19317.4,20345.0)	
7				

You can edit additional properties for the Excel output by double clicking on the Excel command from the report script.

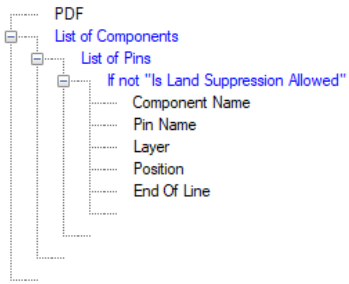
Checking the **Auto Fit Column Width** box will enable the columns to fit the data within it or leave the columns at the default width defined by Excel.



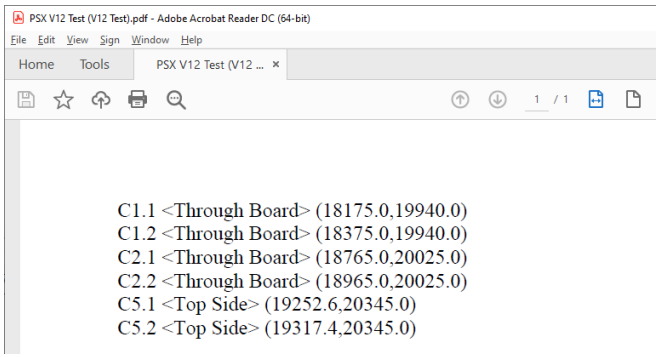
New command - Output to PDF

A new command has been added to allow the creation of PDF based reports.

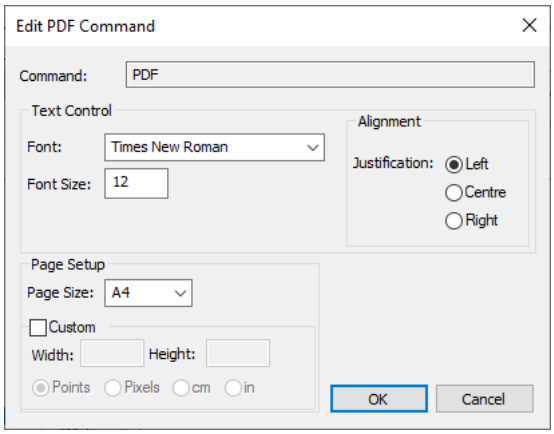
The new **PDF** command defines the report to be written in and saved as a .pdf file.



This will export the report directly and open your PDF viewer, such as Acrobat:



You can edit additional properties for the PDF output by double clicking on the PDF command from the report script.



From here you can change; The font and font size that will be used in the report, the overall text alignment of the document, and the page size, selected from either a list of standard ISO formatted page sizes (A0 – A5) or a custom width and height in either Point, Pixel, centimetre or inch units.

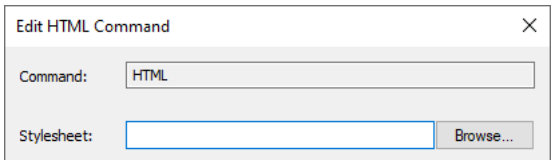
New command - Output to HTML

Four new commands have been added to Report Maker to enable the creation of HTML based reports.

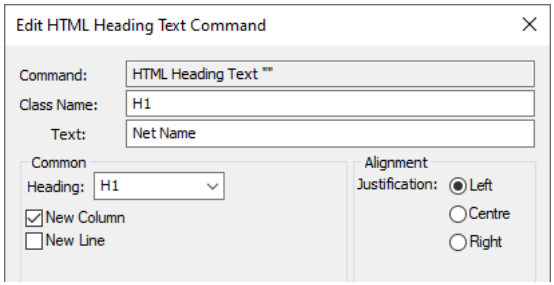
All the commands have additional options which can be edited by double clicking on the command from the report script.

The **HTML** command defines the report to be saved as a .html file. Using this will force the report to have the file extension .html. This should be placed at the start of the format script.

Double clicking on this command, you can browse and define a stylesheet to be used by the other HTML commands throughout the report. This will be a standard formatted HTML CSS (Cascading Style Sheet) file.



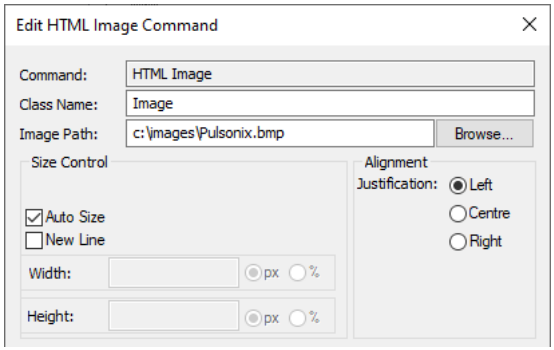
The **HTML Heading Text** command allows you to write HTML headings to the report, supporting (H1 – H6).



From this command you can define; the HTML **Class Name** assigned to this heading (when a CSS Stylesheet is defined in the HTML command), the **Text** that is to be written, whether to write in a

New Column (tables only), whether this command is the last in the line (**New Line** after), the **Heading** type (selected from a drop-down list) and the **Alignment** of the text in the HTML file.

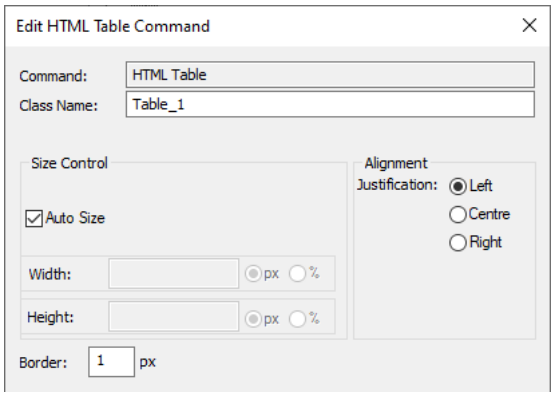
The **HTML Image** command allows you to write an image to an HTML output report. Image formats supported are: JPEG, TIF and BMP



From this command you can define the **Class Name** assigned to this image (when a CSS Stylesheet is defined in the HTML command), the **Path** to the image, the **Size** of the image (either **Auto Size** or manually specified **Width** and **Height** in either pixels or percent units) and the overall **Alignment** of the image in the HTML file.

Note, with **Auto Size** unchecked, values for Width and Height must be greater than zero. If a value is entered into at least one of these fields then the other can be zero, in which case, the image will be size proportionally. For example, if Width is made 80%, and Height of zero (0), then the Height will be 80% also.

The **HTML Table** command allows you to draw a table to an HTML output report.

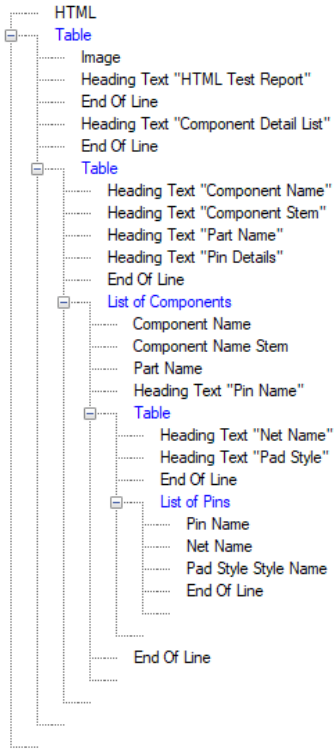


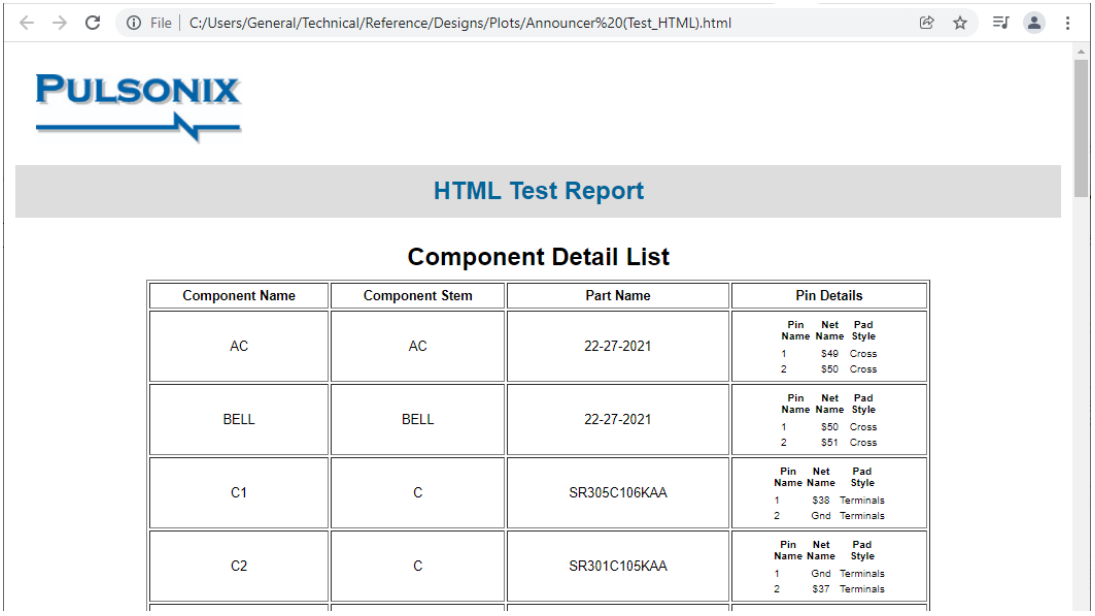
From this command you can define; the **Class Name** assigned to this table (when a CSS Stylesheet is defined in the HTML command), the **Size** of the table (either **Auto Size** or manually specified **Width** and **Height** in either pixels or percent units), the overall **Alignment** of the table and the **Border** thickness of the table (in pixels).

Note, with **Auto Size** unchecked, values for Width and Height must be greater than zero. If a value is entered into at least one of these fields then the other can be zero, in which case, the default value for the other will be used.

Example Report

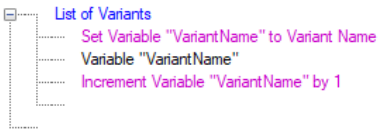
As an example of the possibilities for HTML, the following shows the format script and resultant report in a web browser:





New funciton - Text Variable Increment by 1

A new function for **Text Variables** has been added to the Report Maker. This new function will take the string in the text variable and increment it by one. This can be added to your design as a User Report to display a list of Variants within the design.



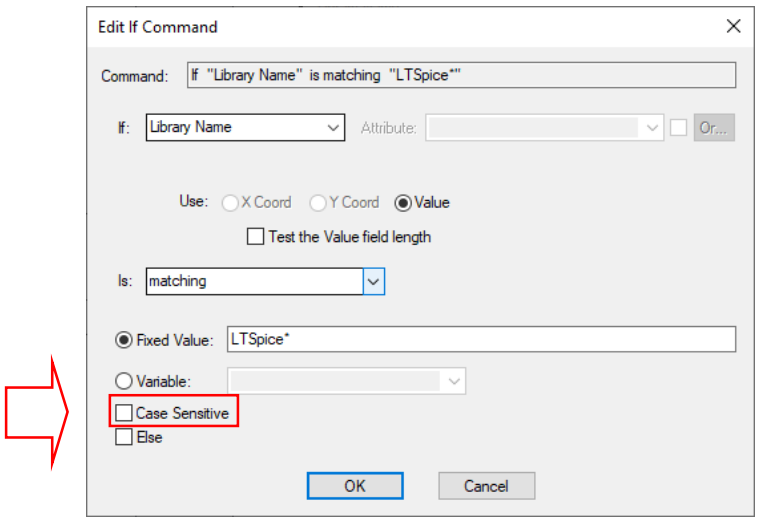
To enable this function, use the **Edit Variable Command** dialog. Assign the variable name to one that is of type **Text**. In the **Action** list there is a new option (radio button) for **Increment Text**.

When enabled, this will increment the number at the end of the text by 1 or if no number is present will add a number to the end of the text starting at 1.

This feature was back-fitted to V11.0

If Command – Case Sensitive option

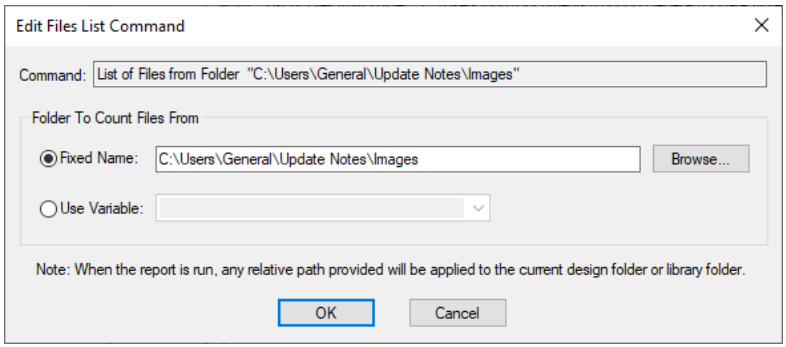
You can now specify whether the comparison in a conditional statement should be case sensitive or not. When editing the **If** command, it will display the **Edit If Command** dialog. There you will see a new check box **Case Sensitive**. If this check box is checked, during relevant comparisons (e.g., when comparing text) it will take character case into consideration. For example, when comparing the word *Hello* and *hello*, it will be false if checked. If the check box is unchecked, then comparing *Hello* and *hello* will be true.



New commands - List of Files and File Name

A new command, **List of Files** has been added to the Report Maker. When the report is run, it will list all the files within a chosen folder.

This feature can be used, for example, in an inserted user report on the PCB design to list all the files generated into the output folders by CAM/Plot. This saves you attempting to copy-paste the plot names into a text item by hand, thus making the new process less error-prone.



The destination can be edited by double clicking the command in the format script section. A folder path can be added by:

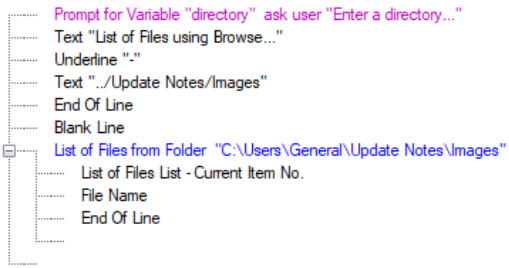
Browsing a folder using the browse button

Typing in a path (this will benefit from relative paths, so folders within the design folder can be selected by just typing in that folder's name).

Reported using a script **Variable**

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.

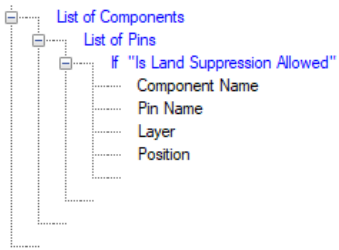
Below is an example script showing how **List of Files** can be used. This includes the use of a **Variable** to ask for user input to define the folder to report and the **File Name** command:



This feature was back-fitted to V11.0 and V10.5

New command - Is Land Suppression Allowed

A new command **Is Land Suppression Allowed,** has been added to support the new Land Suppression override function on a Layer Class (documented above). When added to a report, it will test and return true or false on whether the pad (bond pad or free pad), via or mounting hole has the property **Allow Suppress Lands** enabled.



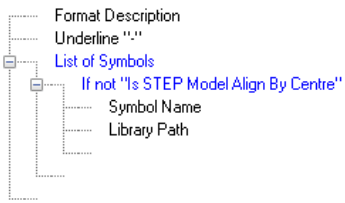
New commands for interrotaing Symbol Library Contents

Attribute In Symbol Library

You can now run the **Attribute In Symbol Library** command from within **List Of Symbols** and **List Of Symbols** to extract attribute information from within the **Schematic Symbols** or **PCB Footprints libraries** in a design.

Is STEP Model Align By Centre

This new command will enable you to test and report if the STEP model has been aligned on a Footprint in the Footprint library by the Centre or Pin 1. This can determine differences of model alignment between the Footprint and the Technology file setting used in the Design.



The test can report a True status for aligned by centre or False if aligned by Pin 1.

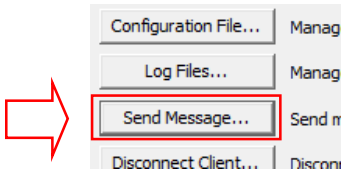
These features were back-fitted to V11.0

Scripting Changes - New LoadTechnology Command

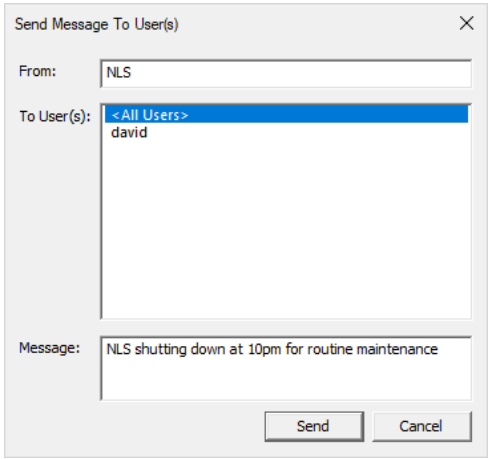
A new **LoadTechnology** function is now available for a **Document** object. Calling this function with a full path to a Technology file name will load that Technology into the design.

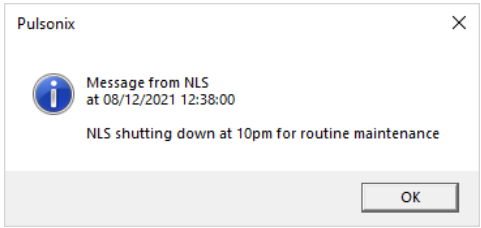
Network License Server (NLS) - Send Message option

In NLSM (the NLS Manager application) there is a new **Send Message** button on the left-hand side:



Selecting this will open the **Send Message To User(s)** dialog:



From here, you can choose one or more users from the list of connected Pulsonix users (or <All Users> to message them all). Type in your message (and optionally the **From** box), and click **Send**.

On the client side, the Pulsonix application will from time to time have a look to see if there are any message pending, and then display them on the screen. These messages are displayed in a popup message box that is separate from the Pulsonix application so it does not block or interfere with the normal operation: