PULSONIX

Pulsonix Design System
V8.5 Update Notes

Copyright Notice

Copyright © WestDev Ltd. 2000-2015
Pulsonix is a Trademark of WestDev Ltd. All rights reserved. E&OE

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

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

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

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

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

Microsoft, Windows, Windows NT and Intellimouse are either registered trademarks or trademarks of Microsoft Corporation.

All other trademarks are acknowledged to their respective owners.

Pulsonix, a division of WestDev Ltd.

Printed in the UK Issue date: 18/02/15 iss 1

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

Support Phone	+44 (0)1684 296 570
Sales Phone	+44 (0)1684 296 551
Fax	+44 (0)1684 296 515
Email	info@pulsonix.com
Web	www.pulsonix.com

Contents

CONTENTS 3
VERSION 8.5 UPDATE SUPPLEMENT 6
Installing the New Version of Pulsonix 6
Licensing 6
Windows XP Support 6
FEATURES CHANGED OR ADDED IN V8.0 7
Import Ariadne (Advanced Import Option) 7
Output LPKF 7
NEW IN VERSION 8.5 8
Panel Editor 8
Overview 8
Creating a New Panel 9
Creating a New Panel using the Panel Wizard 9
The Panel Editor 12
Adding More Detail to the Panel 17
Panel Defaults 20
Report Maker in Panels 23
Panel Checks in Layer Class dialog 23
Rearrange Multiple Items Changes 24
Set Symbol Origin At Centre Of Pads 24
New Enhanced Graphics Mode 24
Display Page Numbers in Design Browser 25
Shortcut Keys availalble on Dockable Bar options 25
Add Part To Design From Lib Manager 25
Drag & Drop To Library Manager 26
Support for Additional External Libraries 27
Suppress Properties on Double-Click action 28
Enable All Libraries 28
New Style Folder Lists 29
Show Group Name on Status Bar 30
Design Tooltips Expanded 31
Draw Current Interactive Grid Only 31
Grids Displayed as X-Crosses 32
Lock Changes 32
Apply Layout Pattern to Multiple Groups 33
Net Names Displayed in Tracks 35
New View Highlighted option 35
Find Bar - Track Styles Changes 36
Reverse Engineer - Place Components To Match PCB 36
Spread Out Option 38
Import Placement CSV 40
Cyclic Design Backups 42
Changes to Folder dialog 45
Specifying CAM Plot and Report File Names and Locations 45
Adding Conditional Report and CAM Plot File Templates 47
Specifying Parts and Symbol Library Report Names and Locations 47

Override Component Pad Position....48
Allow Interactive Reposition for Bond Pads (Chip Packaging Toolkit Option)....49
Copy Tags status in Attribute Editor....49
Position STEP Model....50
Select STEP Filename Model from drop down list....51
Attribute <STEP Filename> available for Doc Symbols....52
Save STEP Models Option....52
PDF Output – Embed Fonts....54
Support for Gerber X2....54
Cross Probe of Star Points....54
Enable Net Styles From Technology Dialog....55
Text Callouts with Net Names....55
Override Exclude Callouts From Plot....58
Preset default Resolve Net Name parameters....58
Additional DRC Checks....59
Net Connectivity Check....59
Panel Item Check....59
Stub Vias Checks....59
Checking of Cross Hatched Copper....60
Staggered Micro-Via spacing....60
Hatch Lines Using Percentage Width and Offset....61
Construction Lines....62
Dividing a Circle....62
Snap to Edge....64
Add Mirrored Line....64
Via In Surface Mount Pads....65
Optimise on Delete....65
No Undo/Redo on Pan/Zoom....66
Layers Bar - Show/Hide Pin Names....66
Find Changes....67
Option to Find in open pages only....67
Find by Block Instance....67
Option to Only Open Last Active Page....68
Report Maker – New Commands....68
Using Reports Folder Template Keywords in Report Maker....69
Changes to the Variable Command....69
Minor Scripting Changes....72
Enhancements to Import Class To Class Spacings CSV....73
Spacing CSV Applied To Layers and Areas....73
Multiple selection....74
Floating Network Licensing – Disconnect Client at Admin Level....74
Technology Files....75
Changes to existing Technology files....75
New Technology files....75
Library Changes....75
New Library Parts Added....75
New Doc Symbols Added....75
Revised Footprint Libraries....75

Version 8.5 Update Supplement

Installing the New Version of Pulsonix

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

To install Pulsonix, insert the CD or double-click on the download executable and wait for a short time. The *Autorun* facility will start the installation procedure. Follow the on-screen commands from the install wizard. You can install Pulsonix 8.5 on top of your existing installation or alongside if you prefer; however, you do not need to uninstall the old version first.

Licensing

Version 8.5 requires a new license. If you are using a version of Pulsonix earlier than Version 8.x, you will require a new license, this 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 and overwrite the existing one. 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. Existing users will be supplied with the new license if they have a valid maintenance contract.

Windows XP Support

Microsoft ended mainstream support for Windows XP in April 2009. In recognition of the widespread use of Windows XP in some regions, Pulsonix has continued to be supported on Windows XP.

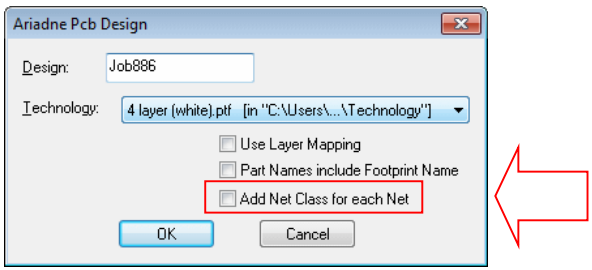
In line with our Support Policy, and bearing in mind that Windows 8 has now been available for more than two years, this will be the last major release of Pulsonix to be supported on Windows XP. More information on support for different versions of Windows can be found on the Support section of our web site under O/S Life Cycles.

Features Changed or Added in V8.0

In the following feature descriptions, you will find some features documented that were included in a late release of 8.0. They have been documented here to highlight them for you.

Import Ariadne (Advanced Import Option)

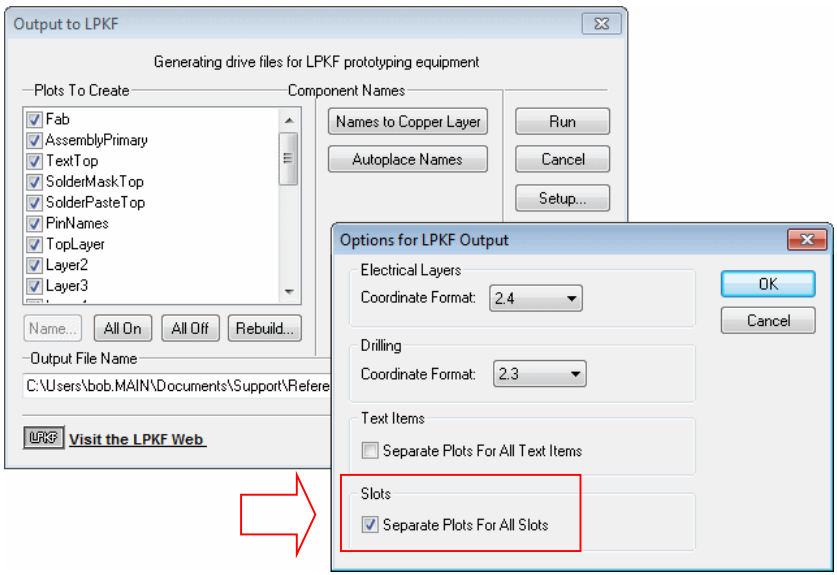
On the **Import Ariadne** dialog, there is a new check box for **adding net classes to each net** that were not originally assigned to a net class in the Ariadne PCB design.



Output LPKF

On the **LPKF Setup** dialog there is a new option **Separate Plots For All Slots**. Slots are non-round holes and can be plotted in two different ways (routed using a NC Drill Excellon-type output or punched).

Check the **Separate Plots For All Slots** to plot plated and unplated slots on their own separate layers within the output file. Leave the box unchecked to include plated slots with the plated drill holes, and to include unplated slots with the unplated drill holes.



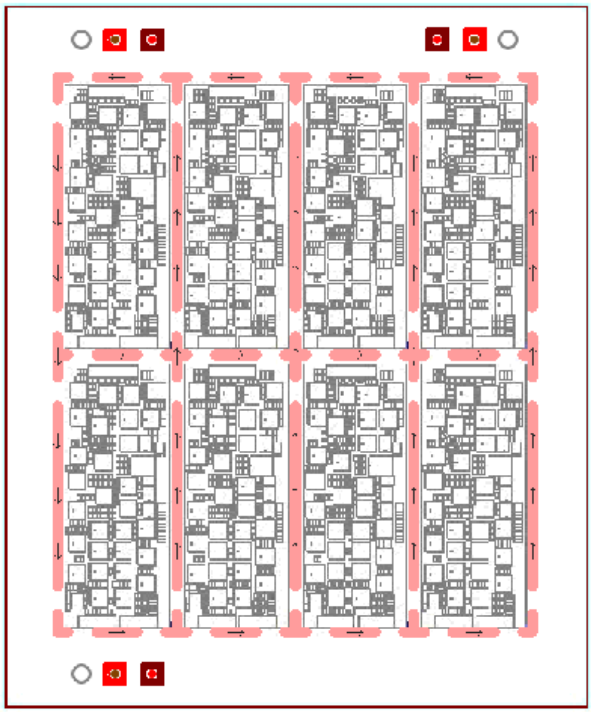
New in Version 8.5

Panel Editor

Overview

Pulsonix 8.5 introduces the **Panel Editor**. This is used post-PCB design editing and enables you to collect PCB designs into a single ‘panel’ and position them ready for plotting. There are tools within this option to create the manufacturing plots in Gerber, Excellon and PDF etc. format using the CAM Plot dialog.

When creating Panels, it is strongly advised that you work with your manufacturer to get the best out of Pulsonix and so you also fully understand their own capabilities and working tolerances.



Panel Layer Visibility

When viewing designs in the Panel Editor, it should be noted that the electrical layers of a panel design are not fully drawn with every detail; instead the board is represented by an infill colour.

What can you do with the Panel Editor?

You can import a single PCB design or multiple PCB designs, then create arrays (called Panelise). You cannot import Gerber files, this editor is used to add Pulsonix PCB designs only. You can add the same PCB design to create multiple copies in an array or you can add multiple different PCB designs to get the most out of your panel. For example, you can add a circular board and then an 'L' shaped board to fit around it.

Once the boards are positioned, shapes can be added for documentation purposes. There are also new shape items for creating tab-routs and V-Score profiling for physically removing the finished PCB from the panel.

Other documentation items can also be added, these might be Bitmaps, drill tables or user reports. A reminder when using Bitmaps; they cannot be plotted to Gerber because of pixel resolution limitations. Free pads and Doc Symbols can be added, these might be used for fiducial markers or panel mounting holes. Doc Symbols might also be used for adding test coupons as well. Text can be added to a panel for documentation purposes, this might be straight text or barcode style text to create a Barcode.

Construction Lines can be added to a panel to 'guide' various items when creating it.

Import options allow Gerber and DXF files to be imported to aid construction of milling paths and tab breakouts.

Additional colour items are available in the **Colour** dialog under the **Others** tab. These are specific to the Panel editor for new panel items.

Before plotting using the **CAM Plots** dialog, the panel can be checked using the panel **DRC** dialog. This has new checks specific to panels and panel functionality.

Creating a New Panel

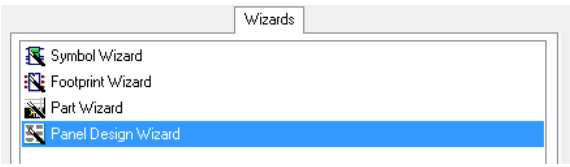
There a two methods for creating a new Panel Design:

- Using the **Panel Wizard** from the **New Wizard** option (**File** menu, **New**, **Wizards**)
- Using the **New Panel Design** option from (**File** menu, **New**, **Designs**)

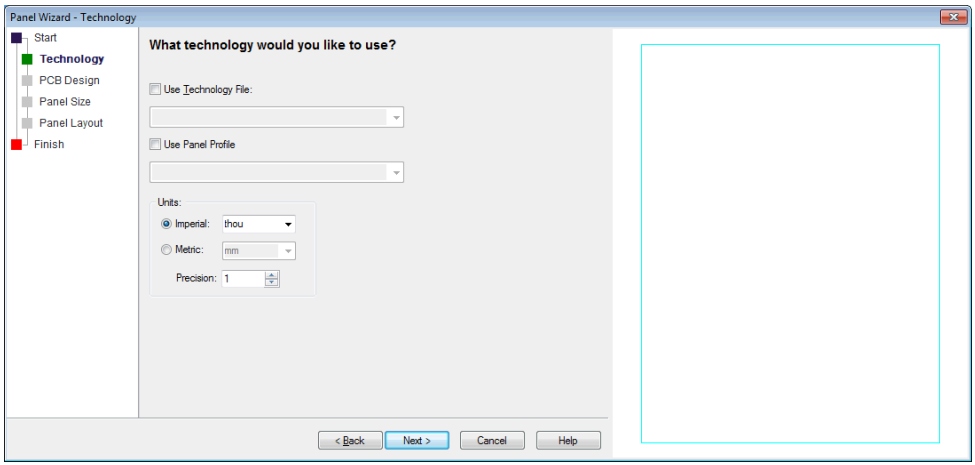
Both methods are described below.

Creating a New Panel using the Panel Wizard

From the **File** menu, **New** option, **Wizards** tab, you can create a new panel using the **Panel Design Wizard**.

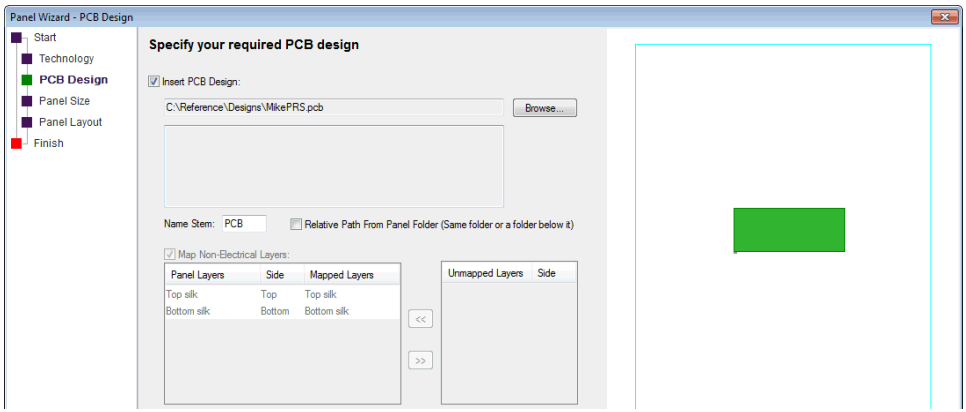


The Panel Wizard guides you through adding **Technology**, the **PCB Design**, panel details, such as **Size** and its **Layout**. A preview window displays your choices as you make them.

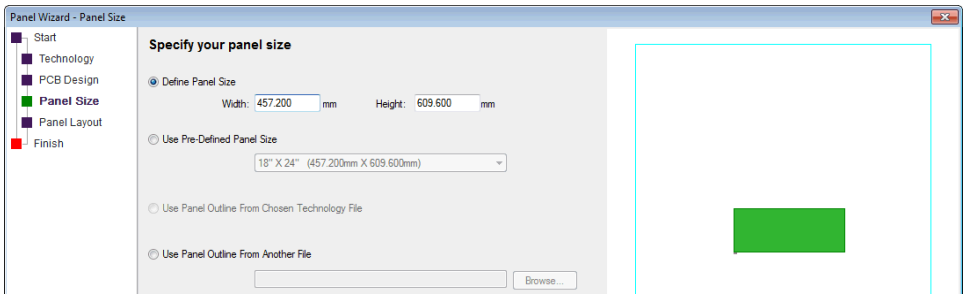


Note: If you don't select a Technology file as this stage, a Technology will be created using the PCB Design on the following tab.

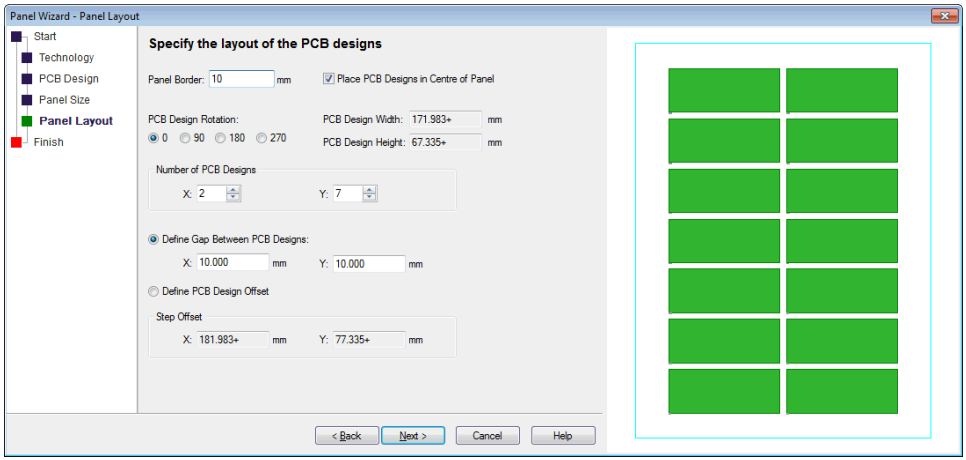
From the **PCB Design** page, choose the design to be used. You can leave this blank if you wish and interactively add the PCB design or designs at a later stage. This dialog also allows you to match mismatched technologies.



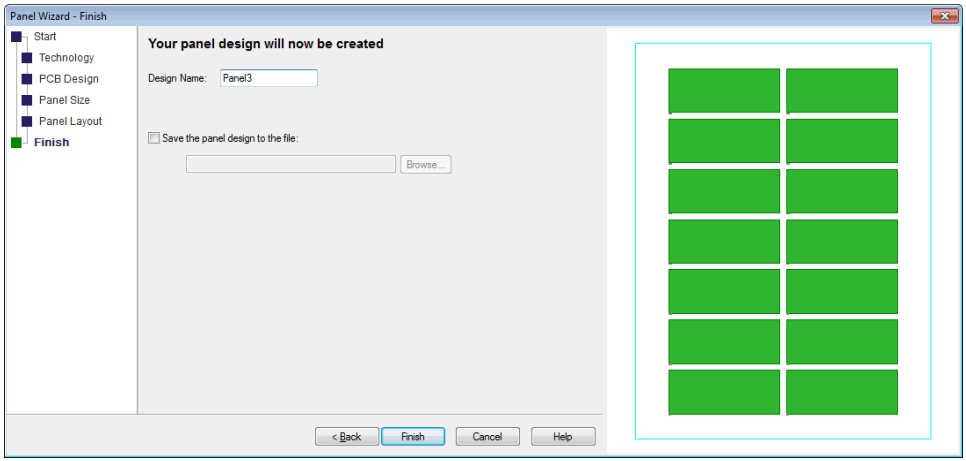
The **Panel Size** dialog enables you to choose the size required. Either type the size or use pre-defined panel sizes from the drop down list available. You can also choose to use an existing panel from a previous design.



The **Panel Layout** page enables you to choose panel parameters, such as the border distance, the number of PCB designs on the panel to create the array and the distance between the designs themselves.



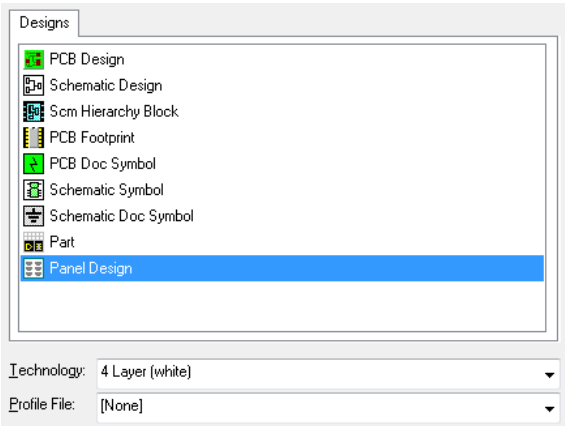
The **Finish** page enables you to create a new panel and start editing it or save it as a panel design and then edit it.



The Panel Editor

Creating a new Panel Design

From the **File** menu, select the **New** option and **Designs**. Select **Panel Design**.



Select the **Technology** file and/or **Profile** file you wish to use. Once **OK** is pressed the **Panel Editor** will open, from there you can add design items to it.

Adding a Panel Outline to the Panel

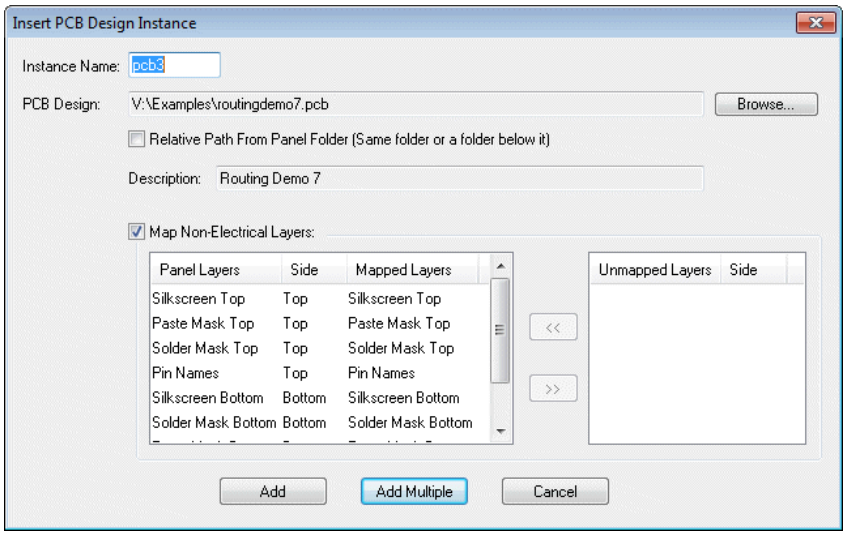
The first operation should be the addition of the actual panel outline. Use the **Insert Panel Outline** option from the **Insert** menu. You may choose a square or rectangle shape and it will always be a closed style. This outline appears on all layers in the design. When adding, use the standard interactive options to create and adjust the size and position (**Type Coordinate** and **Type Offset** etc.).

When zoomed in sufficiently, you will see small directional arrows on the outline, these indicate the routing direction.

If you decide you'd like to change the style of this Panel Outline to another shape style, right click and choose **Change Shape Type** from the context menu. Conversely, you can also add any style of shape to the panel design and convert it to a Panel Outline shape.

Adding PCB Designs to a Panel

Once the Panel Outline has been added, you can use the **Insert PCB Design** option to add Pulsonix PCB designs to it and map its non-electrical layers.



Each PCB instance added must have a unique name, enter the required name in the Instance Name box. These names are only used to uniquely identify each PCB instance in the panel.

Use the **Browse** button to choose the PCB design to place in the panel. Its file path and description will be placed in the dialog information boxes.

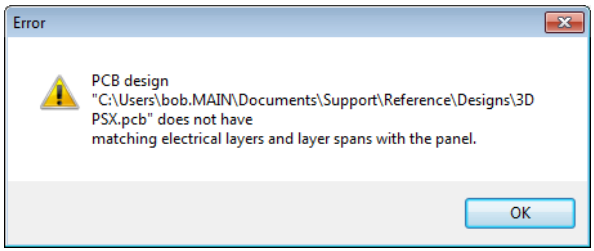
There are two ways of locating the PCB design file in the future when the panel design needs to access its data, for example when generating plots. The first way is to use its absolute file location. The second way is to access it in the same folder as the panel design, or in a folder below the panel design. Do this by checking the **Relative Path From Panel Folder** box. This is useful when saving the panel design with the PCB design, because if they are both then moved to a different folder the panel will still be able to find the design.

The number of physical layers in the PCB design, and the layer span structure must match the Panel Design. If they do not you will get an error message and not be able to add the PCB design. If the physical layers do match you will be given the choice of mapping the names of the non-electrical layers in the PCB design to the layer names in the panel technology data. If you do not map the names yourself, only the non-electrical layers with names that exactly match will be used when creating plots.

To map a layer name, select the PCB layer you want to map in the **Unmapped Layers** column, then select the panel layer you want to map to in the **Panel Layers** column and press the "<<" button. If the sides are the same the layer name should move columns. To remove a layer name mapping select it in the **Mapped Layers** column and press the ">>" button. It will move to the **Unmapped Layers** column. Any non-electrical layers left in the **Unmapped Layers** column will not be used for this PCB design when creating plots from the panel.

Press **Add** to add a single instance of the PCB design, or press **Add Multiple** to enter the **Add Multiple Panel PCB Items** dialog to add an array of PCB designs. You will now be able to position the PCB instance within the panel.

If you attempt to add a PCB Design instance that has mismatched layer items, you are presented with an error message dialog:



You will not be able to add the PCB design to the panel and must correct the layer within that design first.

Editing the PCB Instance

Once added to the panel design the individual PCB instances can be moved, rotated and aligned. By default, the origin of the PCB design is the bottom left corner of the board extents, but you can define a PCB Panel Origin in the PCB design using Place PCB Panel Origin from the context menu. The advantage of using a panel origin is when the PCB design is updated in the panel it will stay at the same place even if its board extents has changed.

Use **Properties** on selected PCB design instances to change the PCB Design file they refer to or to change the layer mapping.

Adding Multiple Designs or Rearranging Multiple Designs

If you wish to add multiple arrays of the same design, you should either start the panel with the **Panel Wizard** or add a **PCB Design Instance**, then use **Replicate** (if only one design has been added) or **Rearrange Multiple Items** if multiple designs have been added.

One the designs(s) have been added and positioned, further detail can be added. See further down this section under *Adding More Detail to the Panel* to see what can be added.

PCB Panel Origins

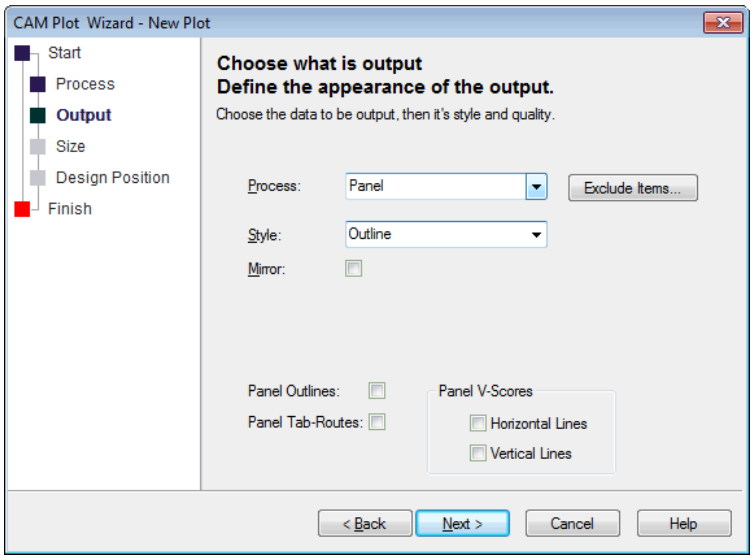
In a PCB design, with nothing selected the context menu contains a new option **Place PCB Panel Origin**. Use this to place an origin in the PCB design that will be used when adding the PCB to a panel design. If it does not exist the lower-left of the board outline bounding box will be used.

This origin has its own colour in the **Colours** dialog, **Others** page.

Exporting and Plotting Panels

As well as using the standard CAM Plot dialog to export panels in Gerber, Excellon, PDF etc. format, you can also extract Reports, create custom reports from the Report Maker option, export to ODB++, generate LPKF ‘plot’s and export to DXF.

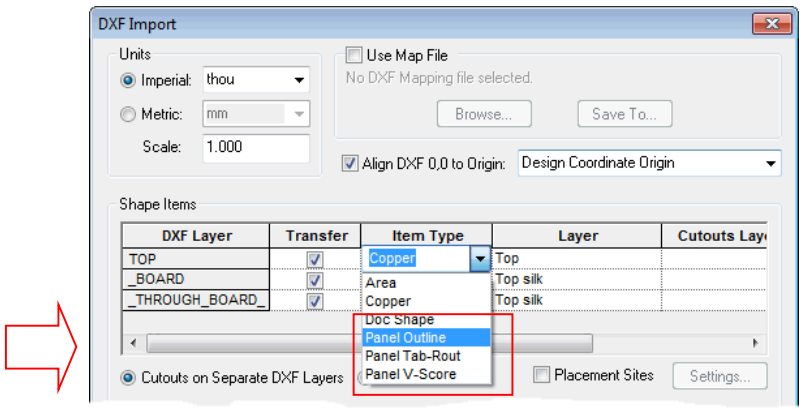
When using the CAM Plot Wizard to create a new plot or edit an existing one, there is a small exception to the standard dialog with the addition on the **Output** page. If **Panel** is selected from the drop down list as the **Process:** you can choose to output specific outputs based on what is required to manufacture the final panel.



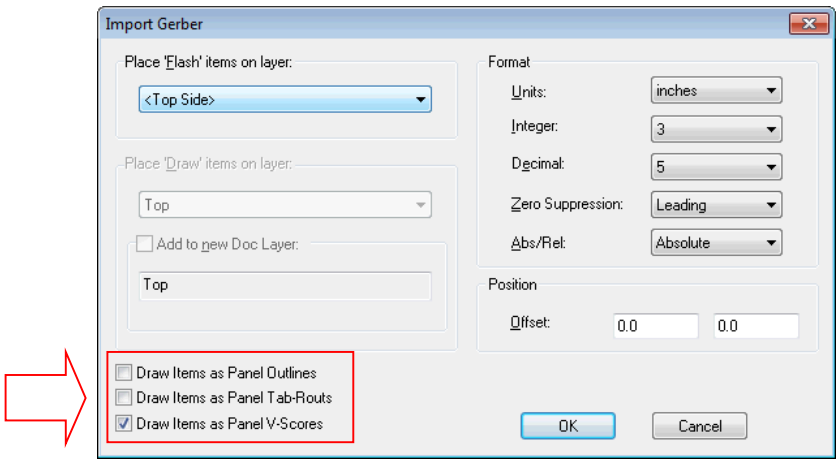
Importing to the Panel Editor

Using the **Import Design Data** option from the **File** menu, you can import **Gerber** and **DXF** files to facilitate the panel creation. Gerber or DXF files may be used for previously created tab paths. DXF files may be used as an alternative to Bitmap for logos (DXF can be plotted to Gerber).

Import DXF – Within this dialog you can specify that DXF layer shapes are imported as one of the special panel shapes (Outline, Tab-Rout or V-Score). Any Tab-Rout shape is acceptable, but only a single panel outline, and only horizontal/vertical single line V-Scores will be created.

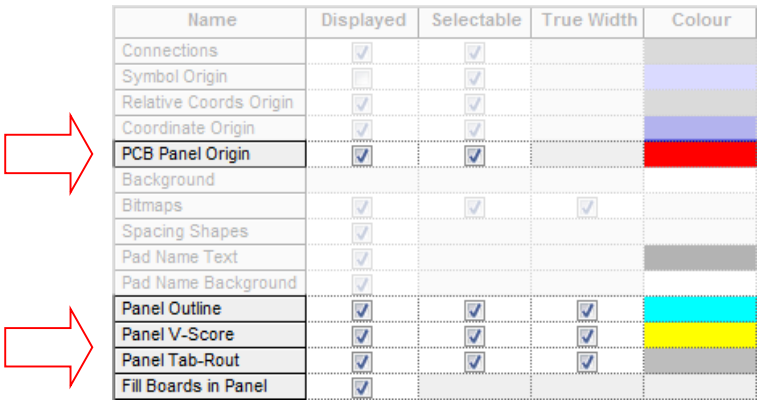


Import Gerber – In the dialog there are new check boxes to choose to read shapes as one of the special panel shapes. Any Tab-Rout shape is acceptable but only a single panel outline, and only horizontal/vertical single line V-Scores will be created.



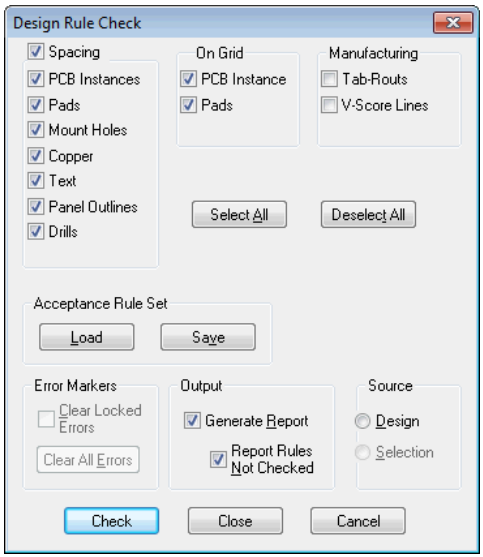
Panel Colours

New colour options have been added to the **Colours** dialog on the **Others** dialog.



Using DRC in the Panel Editor

The **Design Rules Check** dialog has been modified for specific Panel features.



Adding More Detail to the Panel

Additional new features have been added to Pulsonix for use when in the Panel Editor. These are used for specific functions and processes, it is important to choose the correct shape type to match the output you require.

- **Panel Outline** – similar to a board outline but this is a special design item to define the overall panel size. This has been described previously above.
- **Panel Tab-Rout** – once the panel has been manufactured, the boards must be removed from the panel. To do this, the board can either be milled out using an NC Drill machine where the board outline is milled between breakout tabs, or it can be 'snapped' out using cut lines created via the V-Score outline.
- **V-Score Line** – this is a straight line added to the panel to indicate a 'score' line for a scoring machine. Once the panel has been scored, it is physically snapped or broken out of the panel.
- **Breakout tabs** – when a solid Panel Tab Rout has been defined, you can 'cut' it into smaller lines to show the routing path for the milling machine. The small portions of the path that remain in-place to hold the board in the panel after routing are called 'tabs'.
- **Change Shape Direction** – this is available on the context menu for all panel special shapes.

Tab-Rout

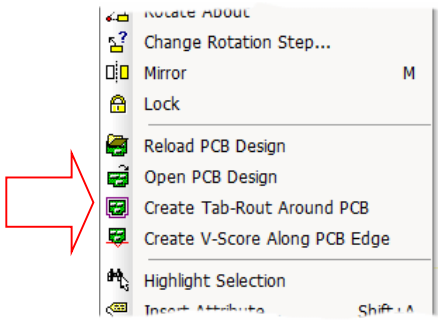
The **Insert Panel Tab-Rout** mode allows you to add the tab-rout shape using one of five shape styles provided: Rectangle, Circle, Triangle, Polygon and Line. Panel tab-routs are created using an unfilled shape which is either closed (non-intersecting and ending at the start point) or an open polyline. These

shapes can be edited after being added to move, add and remove corners, and can have breakout tabs removed from them using the **Create Breakout Tabs** option.

From the **Insert** menu, select the **Insert Panel Tab-Rout** > *shapes* option.

If you decide you'd like to change the style of this Tab-Rout to another shape style, right click and choose **Change Shape Type** from the context menu.

Tab-Routs can also be created by selecting a PCB Instance in a panel design and using the **Create Tab-Rout Around PCB** from the context menu to create a tab-rout around the edge of the selected PCB board outline. The shape will be created so that the inside edge touches the centre line through the board outline, which represents the outside edge of the PCB.



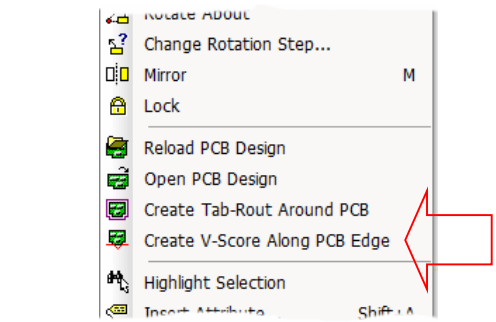
Once added to your panel, this design item type can be output using the **CAM Plot** dialog and selecting its type from the list when **Panel** is selected as the Process.

V-score Line

Panel V-Score lines are created using a single orthogonal line. These lines can be edited after being added to move, rotate, lengthen and shorten them. They cannot be changed from being a single line parallel to the x-axis or y-axis.

From the **Insert** menu, select the **Insert Panel V-score Line** option. This design can only be inserted as line items.

V-Score lines can also be created by selecting an orthogonal board outline in a PCB Instance within a panel design and using the **Create V-Score Along PCB Edge** from the context menu to create a line along the selected PCB board segment.



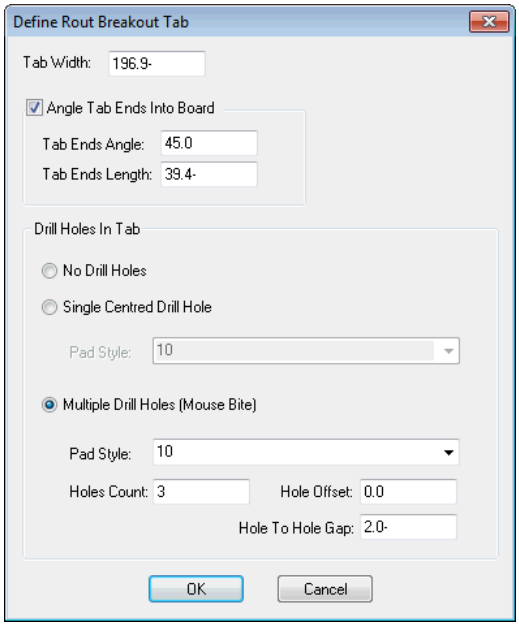
If you decide you'd like to change the style of this V-Score Line to another shape style, right click and choose **Change Shape Type** from the context menu.

Once added to your panel, this design item type can be output using the **CAM Plot** dialog and selecting its type from the list when **Panel** is selected as the Process.

Create Breakout Tabs

The **Create Breakout Tabs** function allows sections of a panel **Tab-Rout** to be removed between two cut lines to create a breakout. Select the **Create Breakout Tabs** option from the **Edit** menu. You will notice the cursor change to the "scissors" cursor.

You can right click and from the context menu, select **Define Rout Breakout Tab** to define the breakout tab parameters:



The current tab width is shown on the status bar at the bottom of the screen. To change it, right click the mouse to use the **Define Rout Breakout Tab** option from the context menu. Type in the required tab width using the current design units.

When using this dialog, the initial default parameters are taken from the **Setup** menu and **Design Settings** option. *For details about the parameters, see the Panel Defaults section below.*

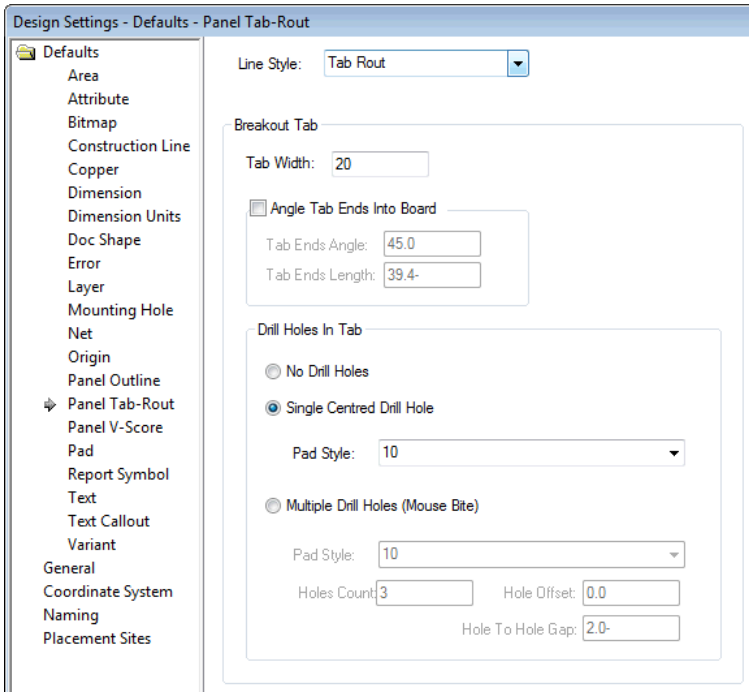
Remove Breakout Tabs

There is also a **Remove Breakout Tabs** option on the **Edit** menu to selectively remove tabs that have been added to the panel.

Panel Defaults

The **Design Settings** option on the **Setup** menu now contains three new Panel defaults:

- Panel Outline
- Panel Tab-Route
- Panel V-Score Line



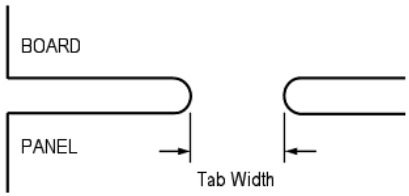
Line Style - choose the line style from the drop down list. This will be used to define the Tab-Rout path in the Panel. Select a line style that has the correct width for the milling tool to be used to cut around the PCB boards in the panel.

Breakout Tab

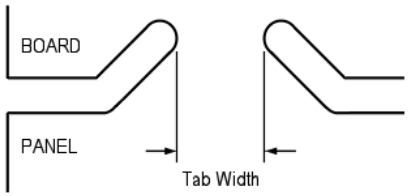
This section defines the parameters to be used when **Create Breakout Tab** is used to create breaks in tab-routs. The PCB board is assumed to be on the left side of each tab-rout segment (looking along the segment in the direction of the arrows drawn on it).

Tab Width - this defines the width of the break in the tab-rout. It is the gap required between the edges of the ends of the tab-rout segments.

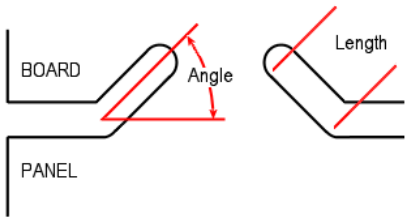
The example below shows the **Tab Width** distance for a straight line breakout tab.



The next example shows how the Tab Width distance is defined when an angle is applied to the tab.



Angle Tab Ends Into Board - Some manufacturers prefer the tab-routs to turn slightly into the board where a breakout tab is required to ensure that the tab breaks inside the board. Check this box if you require angled tab-rout ends. The values define the **Angle** and the **Length** as follows:



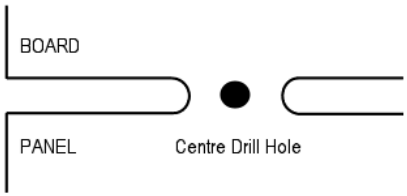
Note: If for some reason you wish the tab to be directed 'outside' of the board outline, use an angle greater than 180 degrees.

Drill Holes In Tab

Drill holes can be added in the breakout tab to aid breaking the PCB boards out. Choose one of the three options.

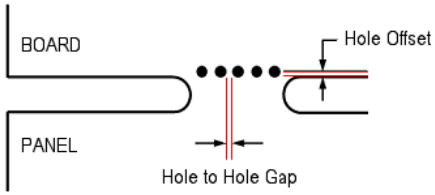
No Drill Holes - Select this if you wish to create a plain breakout tab without any drilled holes.

Single Centred Drill Hole - Select this if you wish to add a single drilled hole to the centre of the tab.



The **Pad Style** is selected from the drop down list and is taken from the Technology file **Pad Styles**. Ideally, you should define a style that is “drill only” with no 'pad' land.

Multiple Drill Holes (Mouse Bite) - Select this if you wish to add multiple drill holes (commonly called mouse bites, rat bites or perforation) to your breakout tab. These "nibble" into the board and ensure that the snap line is within the board when broken out from the panel.



The **Pad Style** is selected from the drop down list and is taken from the Technology file **Pad Styles**. Ideally, you should define a style that is a drill only with no 'pad' land.

Holes Count - use this to define how many holes will be drilled between the breakout tab. The holes will be positioned in the centre of the tab.

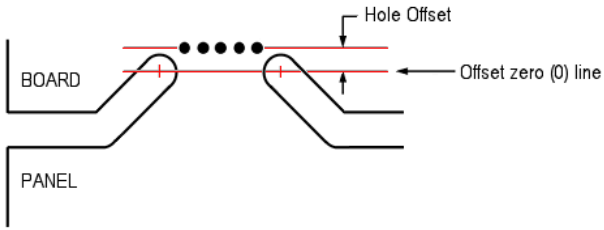
Hole to Hole Gap defines the distance between the drill holes with the spacing taken from the edge of the one drill hole to the edge of the next. Use a combination of hole count, size of drill and hole to hole spacing to ensure the mouse bites span the tab.

Hole Offset is the amount that the drill holes are 'indented' into the board. This value is taken from the inside edge of the tab-rout to the centre of the drill hole. Use this to further ensure the break line in the tab is inside the board. Care should be taken to not move the holes too far into the board where they may clash with items on the PCB.

Note: If you wish the holes to be drilled 'outside' the board outline, use a negative offset value. Normally, the number will be positive.

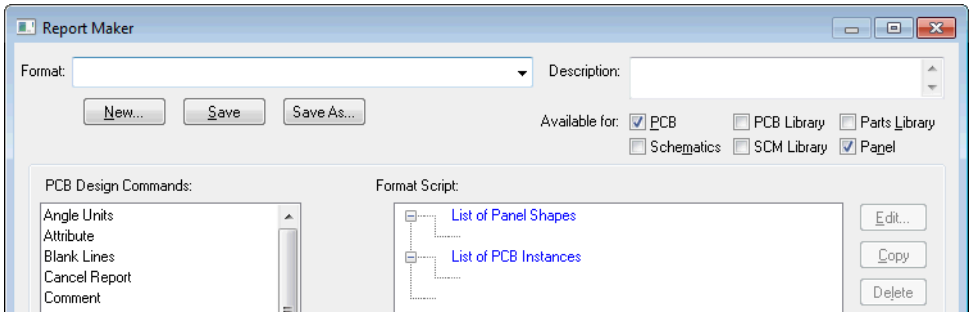
Special Cases

When the tab has an end angle, the **Hole Offset** is now taken from the centre of the drill used for the tab. This is shown in the following example.



Report Maker in Panels

As well as being able to include **Report Maker** files in the **Panel** itself as a **User Report**, the ability to create specific **Panel** reports has been added. The new **Panel** check box enables reports to be run from a panel design.



If you have a report such as a pick and place placement list that can be run from a PCB design, the new top-level command below will enable this to still be run in a Panel Design:

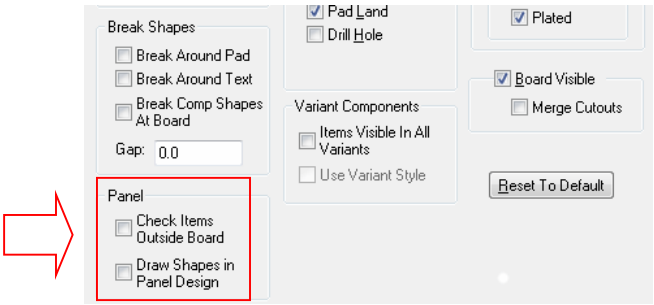
```
Include PCB Instance Contents
```

The Report Maker can now accept the following commands:

```
List of PCB Instances
  Position / Rotation etc.
  PCB Instance Name
  PCB Design Name
  PCB Design Path
List of Panel Shapes
  Type
  Length
  List of Segments
  List of Cutouts
```

Panel Checks in Layer Class dialog

There are two new check boxes in the **Layer Class** dialog to support **Panels**:



Use these check boxes to include items that appear on layers that use this **Layer Class** in **DRC** checks performed using the **Panel Items In Design** check.

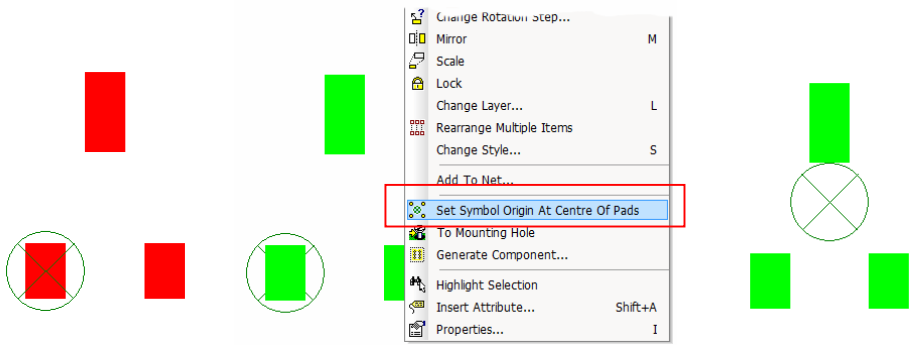
The two Panels check boxes allow **Check Items Outside Board** to be flagged during checking and the ability to **Draw Shapes in Panel Designs** when they appear on a Layer that uses this Layer Class. It means shapes can appear in the design but will not be used if the design is used within a Panel.

Rearrange Multiple Items Changes

The **Rearrange Multiple Items** option now let's you rearrange Pads with different styles and Components with different symbols. This is particularly useful when rearranging pads with different styles in the Footprint Editor or Components in a design such as a selection of resistors and capacitors which you may wish to line up in arrays.

Set Symbol Origin At Centre Of Pads

A new option available from the context menu when editing Symbols or Footprints. **Set Symbol Origin At Centre Of Pads** sets the symbol origin at the centre of the box which includes all positions of all the selected pads and mounting holes. If nothing is selected, it will use all pads and mounting holes. This option is available as a command to be assigned to a shortcut key.



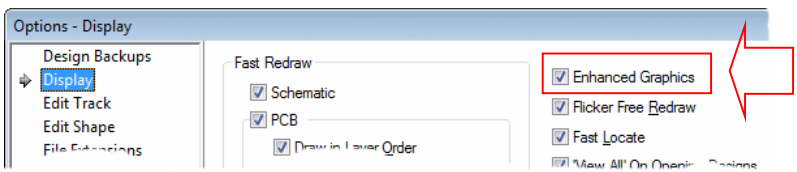
Symbol as initially created with Pin 1 as the Symbol Origin

All pins are selected (or nothing selected), then right click to reveal context menu option. Select **Set Symbol Origin At Centre Of Pads** option

Symbol Origin has now been moved to the centre of the pads

New Enhanced Graphics Mode

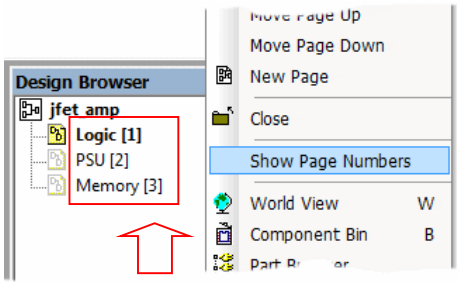
A new option (enabled by default with an 8.5 installation) is available to use **Enhanced Graphics** in Pulsonix. This new mode can be toggled on and off from within the **Options** dialog and **Display**.



You may find that you need update your graphics driver in order to fully utilise this feature.

Display Page Numbers in Design Browser

When using the **Design Browser**, there is a new option on the context menu to **Show Page Numbers** next to the page name. When selected, this will put the page number in square brackets after the page name, as shown in the example below [1], [2] and [3].



Shortcut Keys availalble on Dockable Bar options

Shortcut key commands are available for a selection of dockable bar options to provide fast access during use:

Find Bar:
- Select All Find Items
- Select Visible Find Items
- Refresh Find Bar

Layers Bar:
- All Layers On
- All Layers Off
- This Side On
- This Side Off

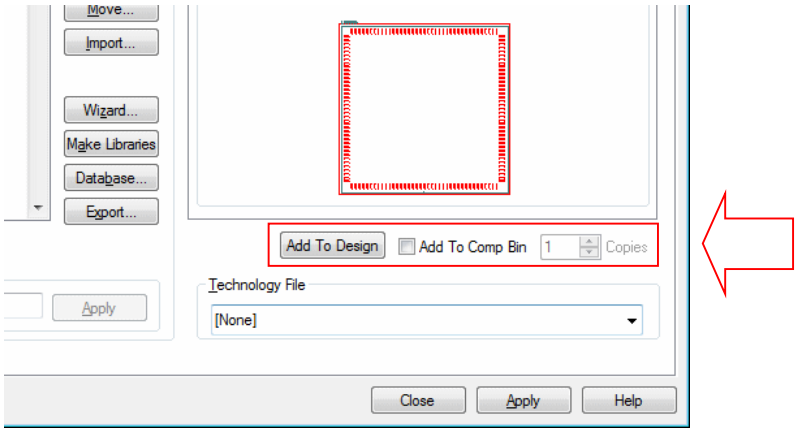
Both Errors Bars:
- Find Error
- Delete Error
- Lock/Unlock Error
- Toggle Sort Order

Component Bin:
- Delete From Bin

Add Part To Design From Lib Manager

When using the **Library Manager** on its **Parts** page, if you have a current PCB or Schematic design open, with a single part name selected from the **Contents:** list, you can interactively add the Part to the design. You can also add a number of copies of the selected Part to the designs' component bin.

Use the **Add To Design** button to insert the Component to the design or add to the Component Bin with this option selected.

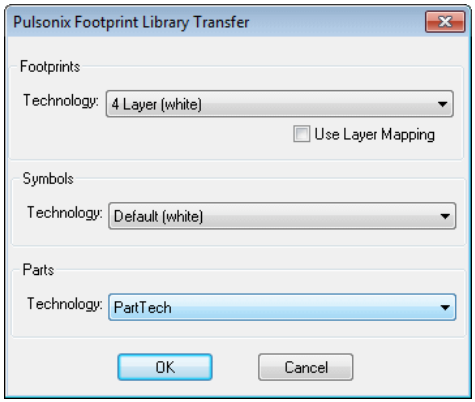


When selecting the **Add To Comp Bin** check box, the **Add** button changes to **Add To Design Comp Bin**.

Drag & Drop To Library Manager

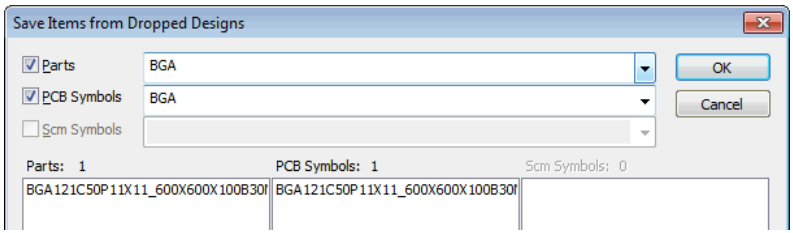
For some file formats, Pulsonix now supports the ability to drag and drop the file onto the Library Manager to create whatever library information it detects in the file. This could be any combination of SCM Symbols, PCB Footprints and Parts. The methodology extends to native Pulsonix binary design as well (.scm and .pcb). Dropping these files onto the Library Manager will open the import manager dialog.

On drop, the following dialog is displayed:



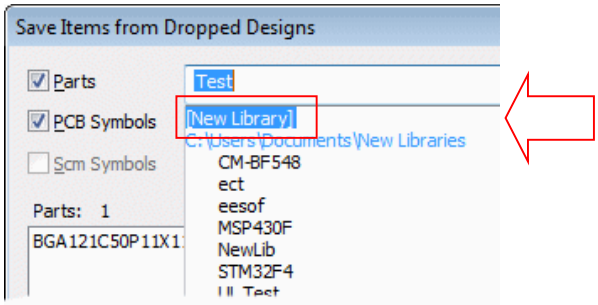
This dialog asks you to confirm the Technology files that will be used during import. Select the required Technology and press **OK**.

You are presented with the **Save Items** dialog:

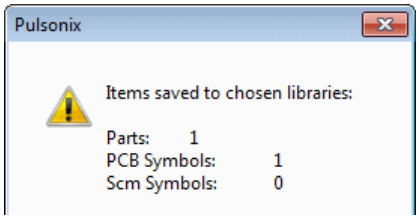


The import dialog will read the contents of the file and determine their type, they will be displayed in the appropriate library list. The import files are capable of containing multiple items as well as individual items.

For each library, either choose the library to import into, or choose **[New Library]** from the drop down list. When you press **OK**, the **Save As** dialog will be displayed into which you can type the new library filename.



Once **OK** is pressed, a confirmation dialog is displayed. Your new library items are now ready to be used.



Support for Additional External Libraries

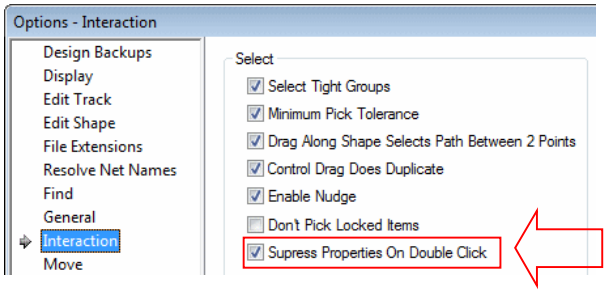
Through its new PLX Native ASCII Format files, Pulsonix can now support additional external libraries available from PCB Libraries (www.pcblibraries.com), Accelerated Designs using their Ultra Librarian product and SnapEDA. Each vendor provides different and unique capabilities of library production in ready-to-use libraries.

Each library supplier will now write .plx format which can be read into the Pulsonix **Library Manager** using the new drag & drop functionality described above.

For more information, please contact your local Pulsonix sales office who will be able to help and advise you further.

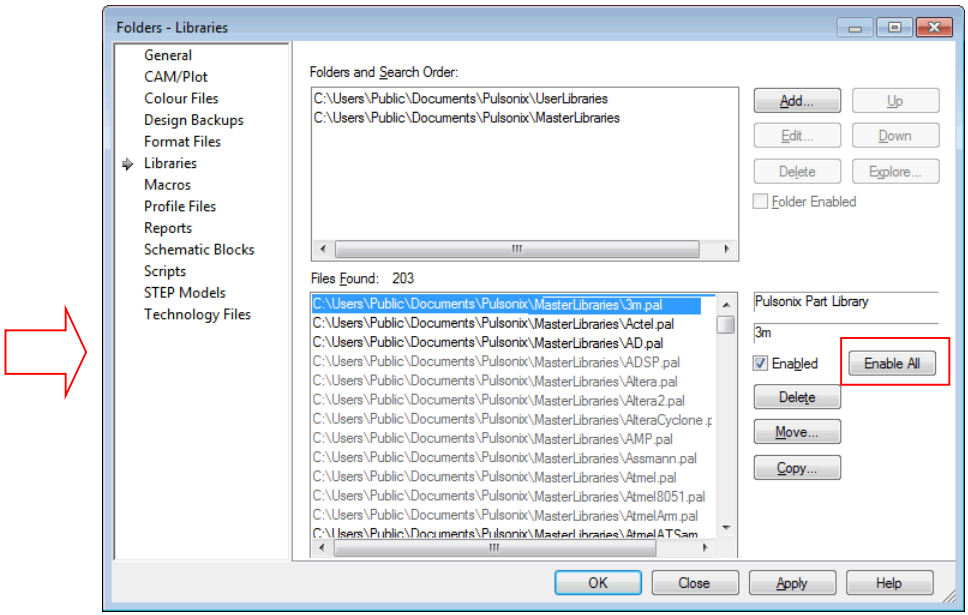
Suppress Properties on Double-Click action

New Options | Interaction option **Suppress Properties on Double Click**. If selected, double click will only edit segments, it will not enter Properties.



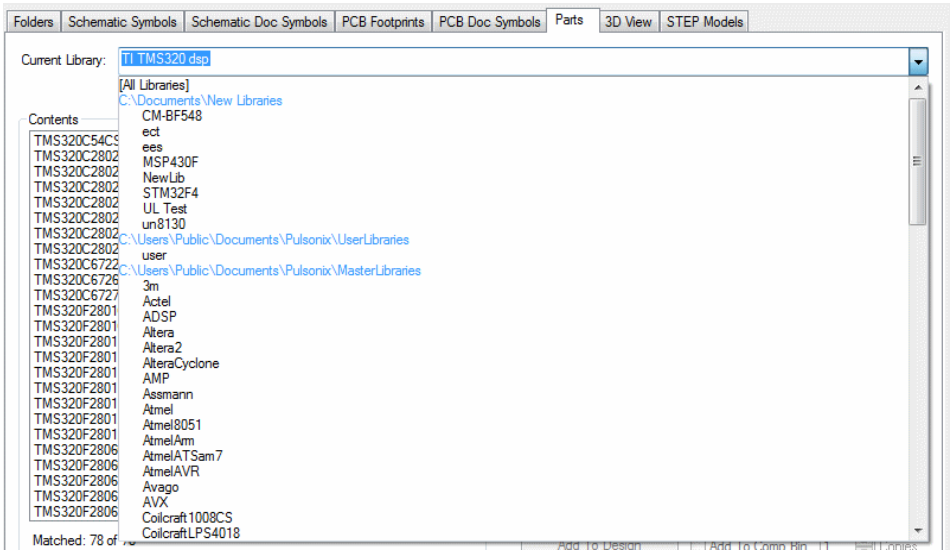
Enable All Libraries

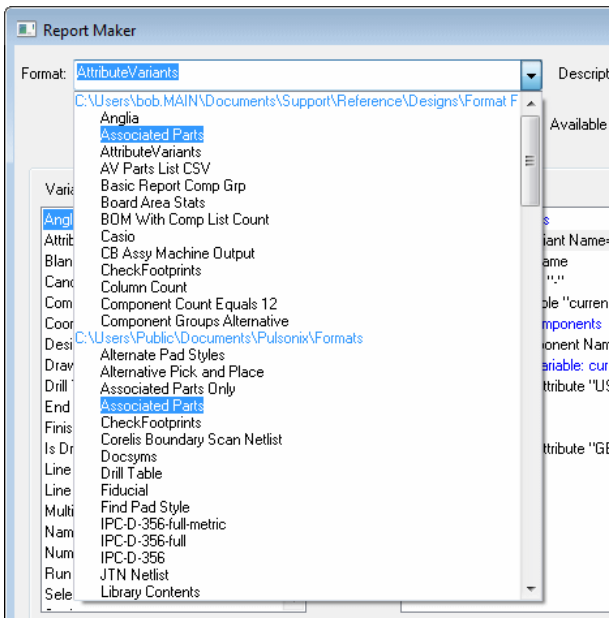
The **Folders** option and **Libraries** page now has an **Enable All** button to enable all libraries found.



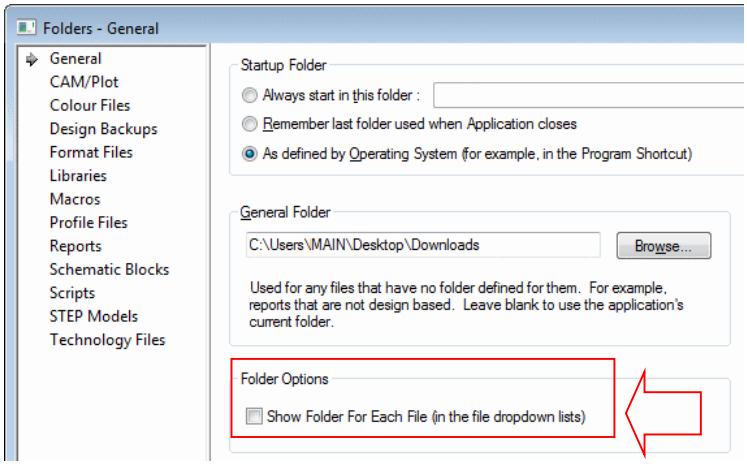
New Style Folder Lists

Where library and folder-based list boxes (such as **Report Maker** selection of **Format Files**) are used, it will now use the new style drop down which displays less information for clarity. This will declutter the list box where multiple folders and multiple files are available.



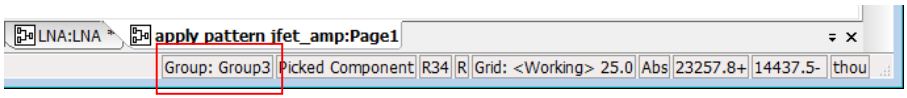


There is a preference to change this to the previous style where the full path is shown. This is available on the **Folders** dialog under **General**. In Version 8.5 the **Show Folder For Each File (in the file dropdown lists)** check box will be unchecked (to use the new feature).

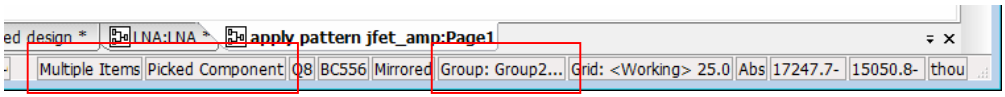


Show Group Name on Status Bar

The group name of the selected items is now shown on the status bar.

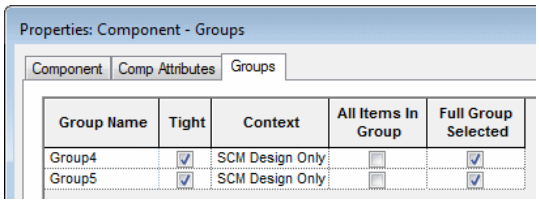


If an entire group (and only that group) is selected then **Multiple Items** is replaced with **Group: <name>**, otherwise the group name is added to the status bar. If the selected items contain more than one group, the main group is shown with three dots after it.



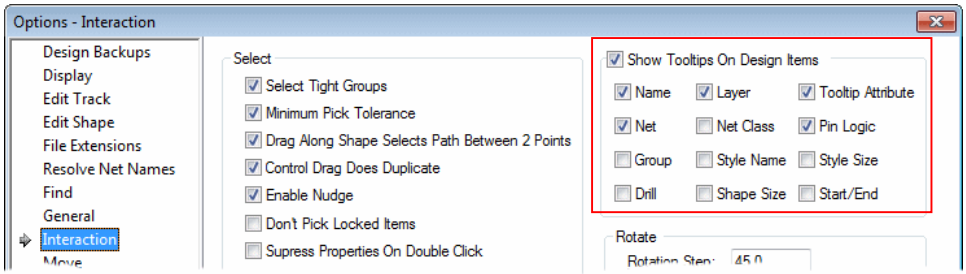
Groups tab in Properties

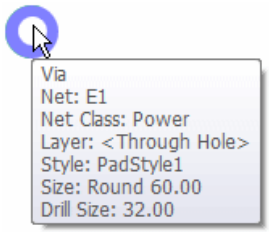
There is now a **Groups** tab in the **Properties** dialog showing which groups the selected items are on. The groups grid is read only and shows if all the selected items are in the group and also if the full group is selected.



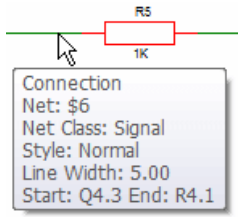
Design Tooltips Expanded

The tooltips selection section now has additional switches to display the item group and other information. This is available in the **Options** dialog under **Interaction** tab. New categories are: Group, Style Name, Style Size, Drill Size, Shape Size and Connection/Track Start End info.





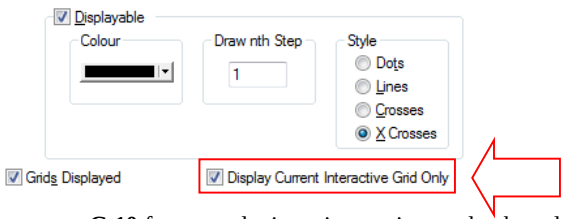
Example above shows a more compete PCB design tooltip



Example above shows a Schematic tooltip example

Draw Current Interactive Grid Only

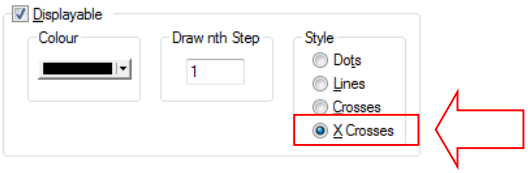
There is now an extra check box on **Grids** dialog, **Draw Current Interactive Grid Only**, so that if checked, any grid being used by an interactive operation will be drawn (even if the particular grid is not displayed). If no grid is currently being used (like in Select mode) then the grids that are marked as **Displayed** will still be displayed.



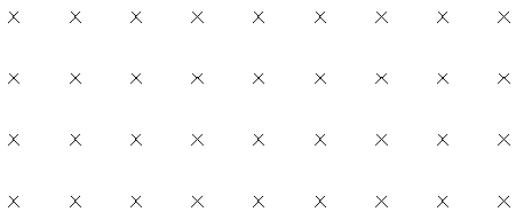
Note, if you type **G 10** for example, in an interactive mode, the colour and style of the grid will be taken from the **Working Grid**.

Grids Displayed as X-Crosses

There is now an extra ‘cross’ style in the **Grids** dialog to draw the grids as X Crosses (small 45-degree cross in the design).



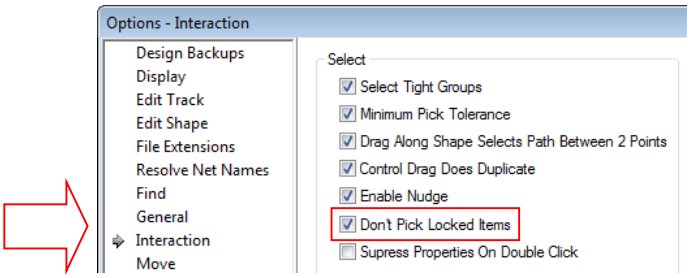
Selecting this option displays the grid in the design like this:



Lock Changes

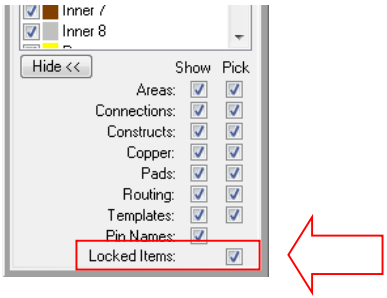
New Interaction - Don’t Pick Locked Items

There is a new option, **Don’t Pick Locked Items**, in the **Options** dialog, **Interaction** tab. If checked, locked items will not be pickable in all circumstances (apart from **Unlock latch mode**). They will be drawn in a dimmed colour to indicate they cannot be picked (the same as if you make a Colours category non-pickable).



Layers Bar Lock Status

The **Picking** of Locked items option is also on the **Layers Browser** bar so that it can be conveniently selected.



Unlock latch mode

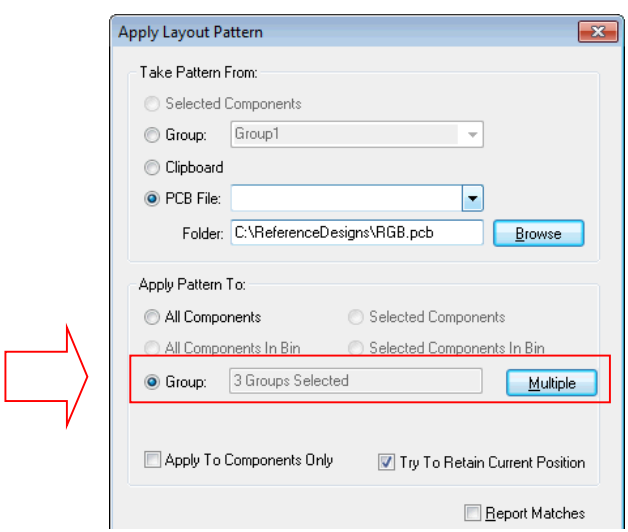
Unlock latch mode will still allow picking of locked items. All locked items will be drawn in the highlight colour to make it easier to pick items to unlock. **View Highlighted** is on the context menu.

Lock and Unlock Track Segment picking

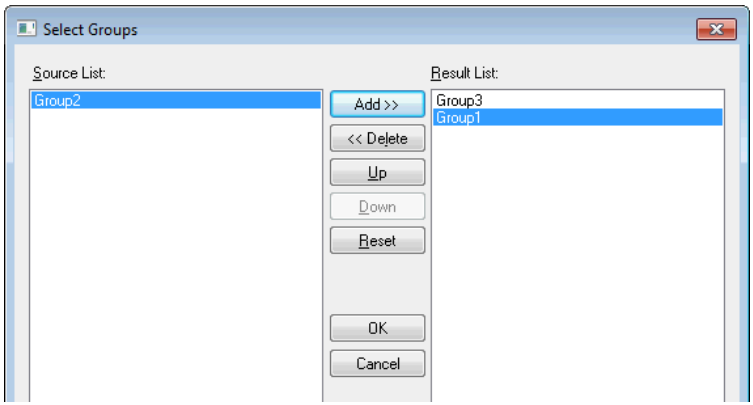
When using **Lock** and **Unlock**, you can now pick track segments (before it was always the whole track selected). Use **Shift-click** to select whole track.

Apply Layout Pattern to Multiple Groups

You can now use the **Apply Layout Pattern** option and apply the pattern to multiple groups which are then laid out in an array pattern. Next to the **Group** button, there is a new **Multiple** button.



Once selected, this opens up the **Select Groups** dialog from where you can choose the group or groups to be used. Once chosen, the group names appear as a selection on the main dialog.

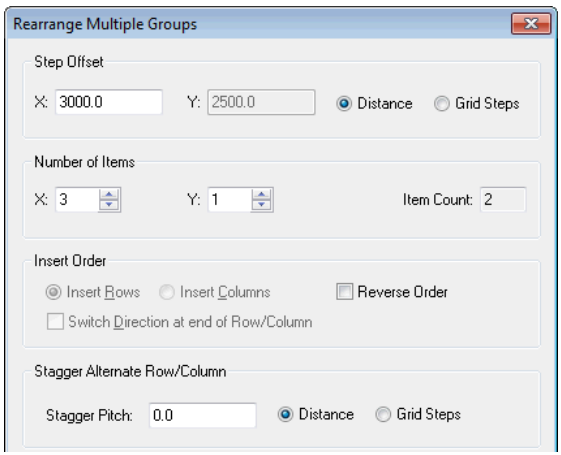


Selecting Multiple Groups

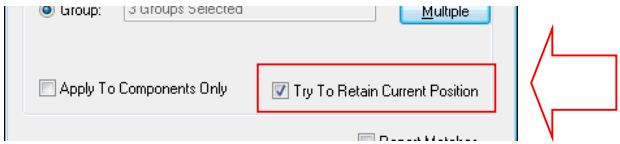
Pressing the **Multiple** button next to the **Group** option, gives you this dialog.

Use the **Add** button to add groups to the **Result List** (or double click on the **Source List**). You can change the order of the Result List using the **Up** and **Down** buttons. The order of the list determines the order they will be arranged.

Once you press **Apply Pattern**, you are prompted to specify the arrangement of the groups. This uses the **Rearrange Multiple Items** dialog, which gives you a comprehensive set of arrangement options.



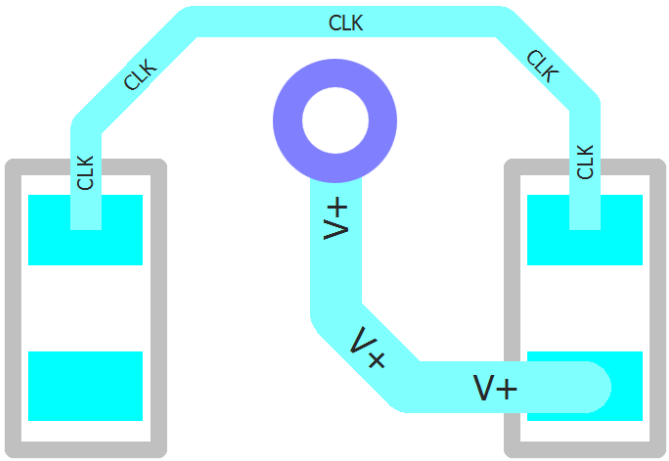
Try To Retain Current Position



Check this option to apply the new pattern to the current pattern but without moving the current pattern position. This is useful where a change to the ‘master’ pattern has been made and you wish to re-apply it to the current pattern(s) without moving them.

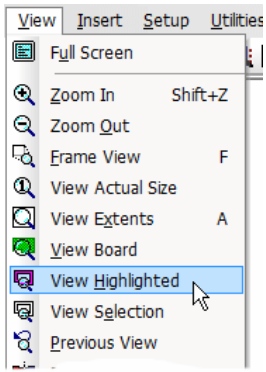
Net Names Displayed in Tracks

Net Names can now be displayed in tracks. There are new switches in the **Options** dialog and **Display** to make various selections and text style choices.



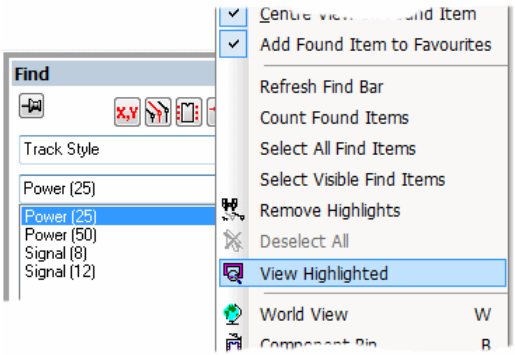
New View Highlighted option

There is a new **View Highlighted** option on **View** menu. This is an alternative method to **View Selection**.



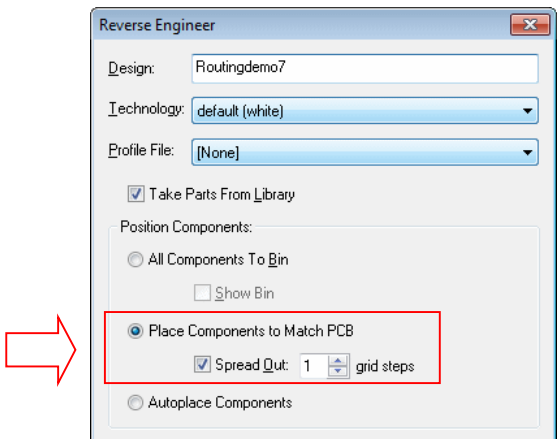
Find Bar - Track Styles Changes

On the **Find Bar**, **Track Styles** will now highlight just the track segments using styles. The **View Highlighted** option is available on the context menu.

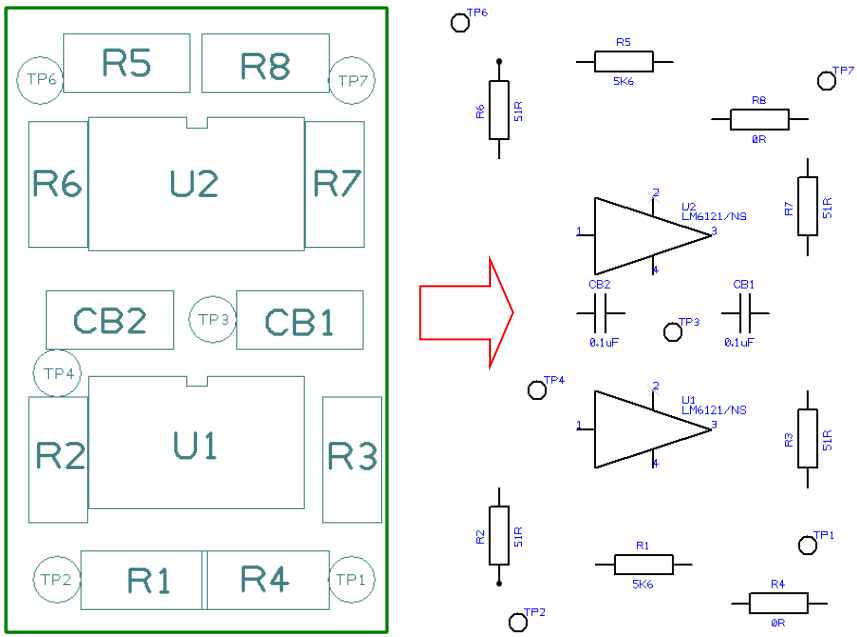


Reverse Engineer - Place Components To Match PCB

This option now has a new component placement option **Place Components To Match PCB**.



Use **Place Components To Match PCB** to place the centre of each schematic gate in the exact same position as the centre of the corresponding Component in the PCB design. If a schematic Component has multiple gates, they will be placed side by side, centred at that position.



The above illustration shows a small sample PCB before, and the Schematic design after the use of **Reverse Engineer** and **Place Components To Match PCB** with the **Spread Out** option selected.

With **Place Components To Match PCB** selected, check the **Spread Out** box to move the schematic gates on to the Component Grid and spread them out so that they have a number of grid steps of space around them. This is also available as an interactive feature in its own right, *see below for more details.*

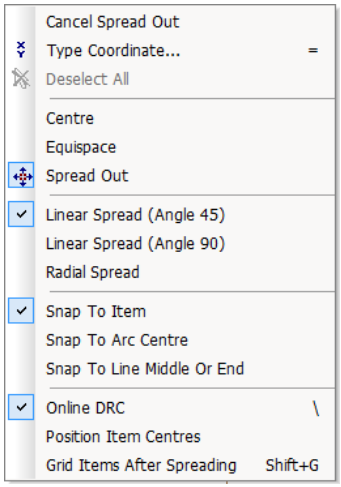
Note: Two pin components will be automatically mirrored and/or rotated to have the same orientation as the PCB Component.

After placement, the Components will be moved to the centre of the working area, any components still off the working area will be placed in the Component Bin.

Spread Out Option

There is a new **Spread Out** option to provide additional interactive tools for placement of items. The **Spread Out** option on the **Utilities** menu provides the ability to centre items between two points; **Equispace** items between two points and **Spread Out** items from an anchor point.

When using Spread Out, a context menu is available to enable various spreading modes:



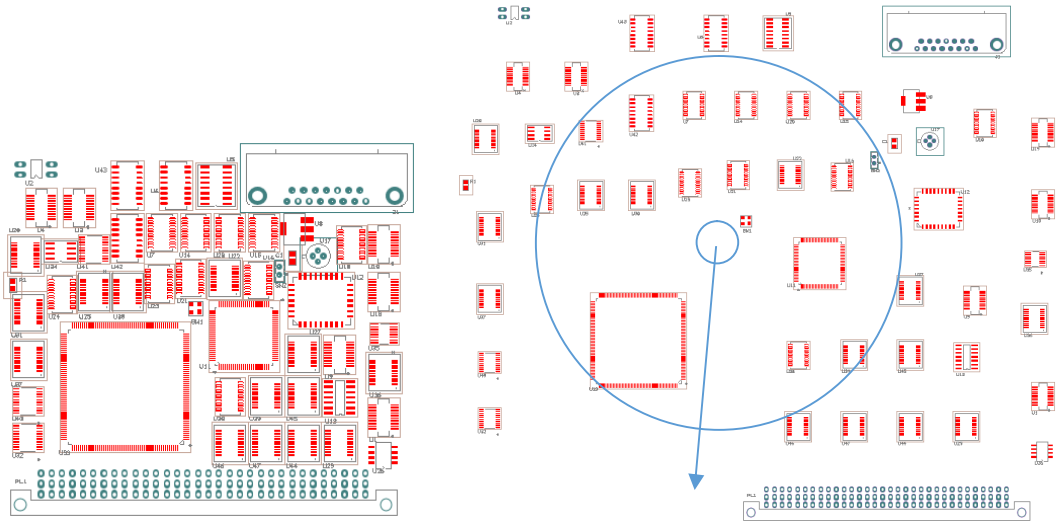
Using the Spread Out option

Use this interactive operation to spread out selected items in various ways to create space in the design. You can select the items to spread out before or after starting the function.

There are four phases to the spreading operation:

- Select the whole items, shape segments or track segments you wish to spread out.
- Choose one of the three methods of spreading that can be used from the context menu.
  - **Centre** - Use this to centre the selection between two points or items.
  - **Equispace** - Use this to equally spread the selected items between two points or items.
  - **Spread Out** - Use this to proportionally spread the selected items away from an anchor point, radially or in a specified direction.
- Click to define an anchor point where you want to spread out from.
- Drag to spread out the items until they are in their required positions, or click to quickly define the second point to spread between.

Below shows the effects of using **Spread Out** on a previously place PCB.



The picture to the left shows the ‘before’ and the picture to the right shows the effect after Spread Out from the Centre has been applied.

The Spreading Methods

The spreading method and direction affect how the spreading is performed. Each method is described here.

Centre

Use this to centre an item, or multiple items, between two points or picked items. For example to centre a selected track segment between two picked pads.

If the spread direction is restricted to **Linear Spread (Angle 90)** the line between the anchor point and end point will be snapped to the closest 90 degree angle and the centre of the selected items will be aligned with the centre of this line. If **Linear Spread (Angle 45)** is used the line will be snapped to the closest 45 degree angle, allowing centring in a 45 degree direction. Using **Radial Spread** will centre the selected items at the mid-point between the anchor point and end point.

If **Align** is enabled from the context menu, the centre of the selection will be moved onto the line between the two picked points. If left unchecked the selection centre point will maintain its original perpendicular distance from this line.

If you have picked items to centre between, and they are not part of the selection, enable the **Between Item Edges** option from the context menu to centre the selection between the closest edges of the picked items, otherwise the origin of the picked items will be used. This option is useful when placing a track segment in the middle of the gap between a large and small pad.

Equispace

Use this to equally space multiple items between two points or two picked items. For example to equally space some parallel track segments between two picked pads.

If the spread direction is restricted to **Linear Spread (Angle 90)** the line between the anchor point and end point will be snapped to the closest 90 degree angle and the selected items will be spread equally in the direction of this line. If **Linear Spread (Angle 45)** is used the line will be snapped to the closest 45 degree angle, allowing spreading in a 45 degree direction. Using **Radial Spread** will first centre the selected items relative to their original positions at the mid-point between the anchor point and end point, and then equally spread them out along that line.

If **Align** is enabled from the context menu, each item will be moved onto the line between the two picked points. If left unchecked the items will maintain their original perpendicular distance from this line.

If you have picked items to equispace between, and they are not part of the selection, enable the **Between Item Edges** option from the context menu to equally spread the selection between the closest edges of the picked items, otherwise the origin of the picked items will be used. This option is useful when equally spreading several track segments in the gap between a large and small pad.

Disable **Equispace Outer Items** on the context menu to specify that the outermost selected items are to be placed at the limits of the line, and the rest of the selected items equispaced between them. For example, use this to frame select five parallel track segments and have the inner three equally spaced between the outer two. Enable **Equispace Outer Items** to include a space between the line ends and the outer selected items.

If **Grid Items After Spreading** is enabled, and the spread direction is a multiple of 90 or 45 degrees, an attempt will be made to keep the gaps equal by gridding them and moving the end items in a little so that the gaps are equal and all items are on grid.

Spread Out

Use this to proportionally spread out the selected items away from an anchor point. For example to spread out some components to make space for tracks between them.

If the spread direction is restricted to **Linear Spread (Angle 90)** the line between the anchor point and end point will be snapped to the closest 90 degree angle and the selected items will be spread in parallel to that line. If **Linear Spread (Angle 45)** is used the line will be snapped to the closest 45 degree angle, allowing spreading in a 45 degree direction. Using **Radial Spread** will spread each selected item radially away from the anchor point.

Whilst dragging a selected item or point, it moves proportionally away from the anchor point in the spread direction. The status bar shows the scale representing this proportion. For example, if the scale is 2.0, all items have been moved twice their original distance from the anchor point.

Import Placement CSV

The PCB Design Editor allows the import of component data in CSV format. Use this import to create the basics of a design using placement information such as position and side, or to move existing components to critical locations based on information from another system.

This is available from the **File** menu and **Import Design Data** option

Component CSV Data

The file must be in ASCII format and must have the file extension ".csv". The first row of the CSV file must contain the column headings for the fields, so that each column of data can be correctly identified. As a minimum, the file must contain a header line with Ref Name, Part Name, X and Y.

The full list of columns that can be imported is: Ref Name,Part Name,Side,Placement Centre X,Placement Centre Y,Rotation.

Some of these can be identified in more than one way, to offer more flexibility in reading files from a range of other sources which may use different terminology to that normally used within Pulsonix:

- Ref Name: Ref, RefDes, Comp Name, Component Name, CompName
- Part Name: Part, PartName, Part Name
- Side: Side, Mirrored, Mirror
- Rotation: Rot, Rotation, Angle
- X, Y: any heading containing an X or a Y

If the Part Name field contains spaces, the field must be enclosed in double quotes " ", e.g. "R 0.063W SMTF 9K".

Values allowed for **Side** are: T[op] or B[ottom]. If the column defines a mirrored state instead, the values allowed are: Y[es] and N[o]

Reading the file

If a Ref Name matches a component that is already in your design, then that component will be moved to the location specified in your CSV file.

If instead there is no component already in the design using that name, the Part Name will be used to add a new component to the design from your Pulsonix library.

Empty values in the X & Y columns means the components are 'unplaced' and those components will be put into the Component Bin.

Values defined for Placement Centre X and Y will be used to place the origin of the component at that location.

Example

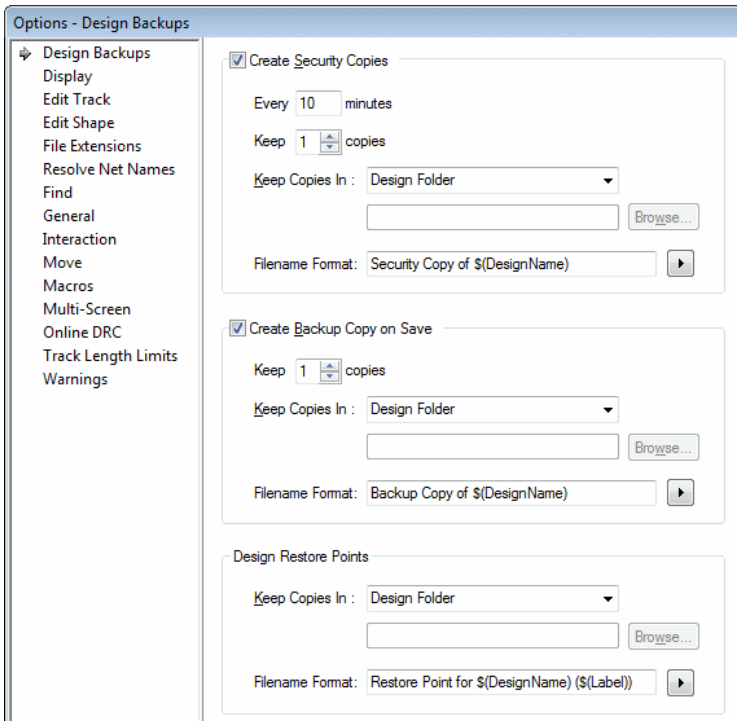
An example of a full Component Position CSV file is shown below:

```
Ref,Part Name,Side,Placement Centre X,Placement Centre Y,Rotation
C1,"c",,Top,18700.0,21800.0,270.0
R1,"R 0.063W SMTF 9K",,Top,19100.0,22100.0,0.0
R2,"R 0.063W SMTF 9K",,Top,19300.0,22100.0,90.0
R3,"R 0.063W SMTF 9K",,Top,19600.0,22100.0,0.0
U1,"SN74AC00D",,Top,19300.0,21700.0,0.0
```

Cyclic Design Backups

Options dialog

Design Backups now have their own section in the **Options** dialog, previously this was on the General tab.



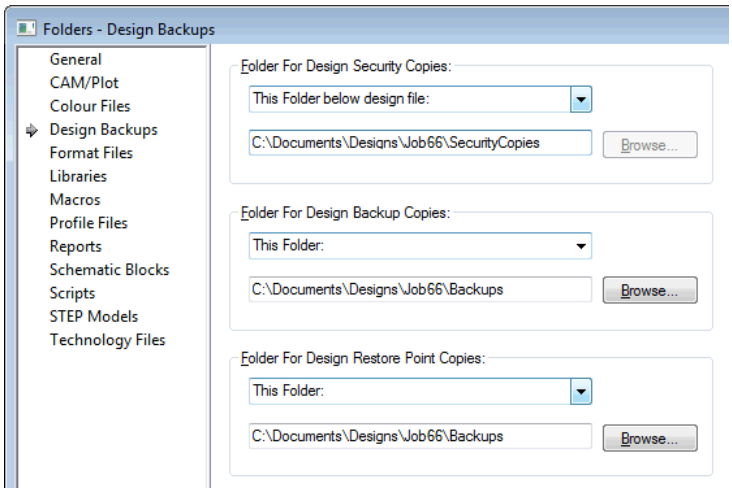
New functionality for design backups is discussed below:

Multiple backup copies

On the **Design Backups** page you can now specify the number of copies that **Security Copies** and **Backup Copy On Save** will save before deleting older ones.

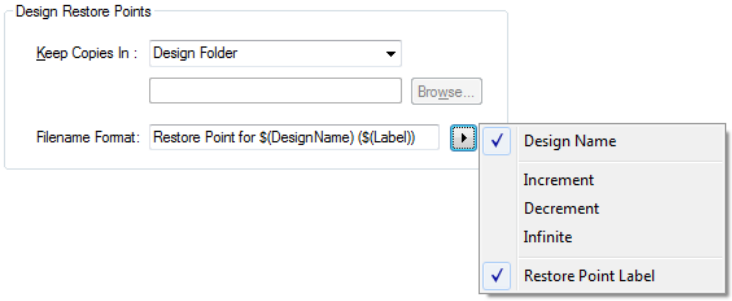
Folders for Backups

You can now define separate folders for each backup type. This can be done in the **Design Backups** page, or in the **Folders** dialog.



User defined format of backup file names

You can create user-defined formats of the backup name by typing fixed text and inserting tags that will be replaced a when a backup name is generated.



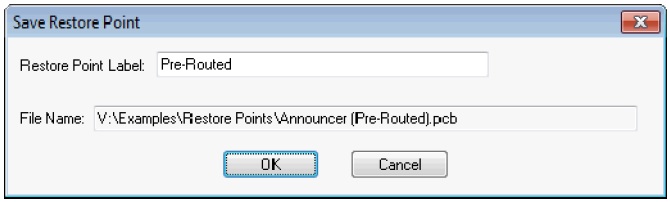
Tags are defined as the **design name** plus one of three numbering schemes – **Increment**, **Decrement** and **Infinite**.

So for example, if number of copies is 3, Increment will produce copies 1 2 3 where 3 is the latest, Decrement - 3 2 1 where 1 is the latest, and Infinite 12 13 14 where 14 is the latest, i.e. it just keeps getting higher.

For the **Design Restore Points** section, you can also select **Restore Point Label** from the context menu.

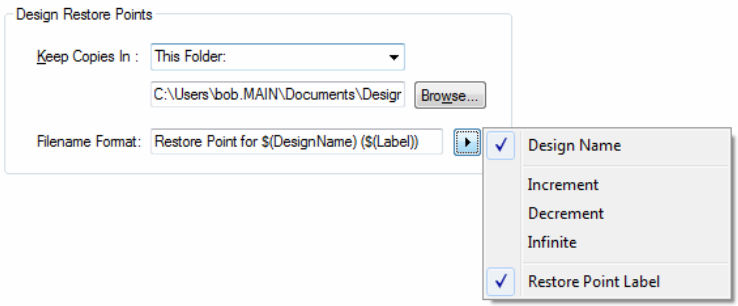
Save and Load Restore Points

There is a new type of backup that is saved on request, called a **Restore Point**, it is used for saving a design that is in a known good state before starting on a session of changes. Save the Restore Point using the **Save Restore Point…** option from the **File** menu. A dialog is displayed:



The saved backup is restored using a new option on the **File** menu named **Restore Backup...**.

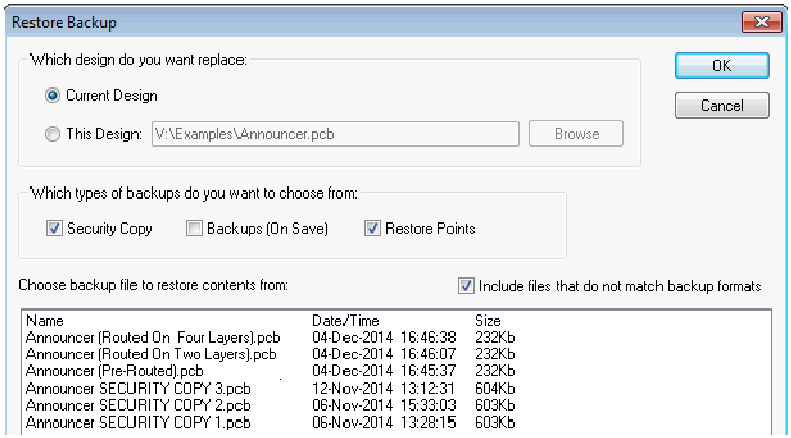
From within the **Options** dialog and **Design Backups** page, you must provide a label for the restore point that will be inserted into its name. For this to happen, the name format must contain the **label** tag.



For example if the name format was “$(DesignName) - $(Label)” then the user could create a restore point for design “MyDesign.Pcb” after autoplace with the label “Before Routing”, so the backup file will be called “MyDesign – Before Routing.pcb”.

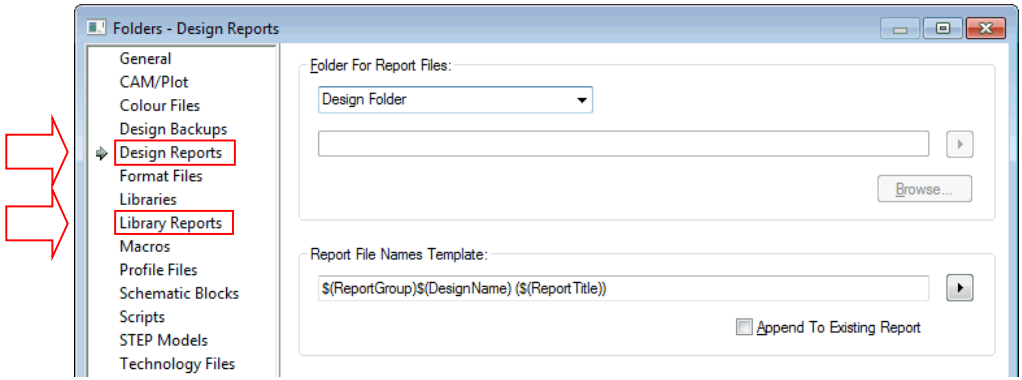
Restore Backup

There is a new option on **File** menu named **Restore Backup...** to restore the contents of a design to be the same as a chosen backup file. This can be performed on the current design, or you can choose the design to have its contents restored from. When selected, this displays a list of backup files to choose from in the dialog.



Changes to Folder dialog

The **Reports** entry in the **Folders** dialog has been modified and enhanced. Reports is now **Design Reports** and a **Library Reports** page has been added. This will enable reports of the different types to be specified.



The formatting of these two dialogs using **Report Templates** is discussed in the immediate section below.

Specifying CAM Plot and Report File Names and Locations

You can now control where **Reports** and **CAM Plots** are written and the format of their file names.

CAM/Plot

Rather than checking boxes to say what is included in the CAM Plot filename, you can now define the filename and folder name using keywords that will be replaced with the relevant text when a plot file is created. This allows you, for example, to use the plot group name to separate plots that are project required as opposed to simple documentation.

When using the **Plot Settings** page in **CAM Plots**, the **Output File Names Based On** section which previously had check boxes, has been now replaced with an **Output File Names Template** section that now has a template edit box and a popup menu button to make the name creation process more convenient when adding keywords. The popup menu button next to the folder edit box is used to make naming selections.

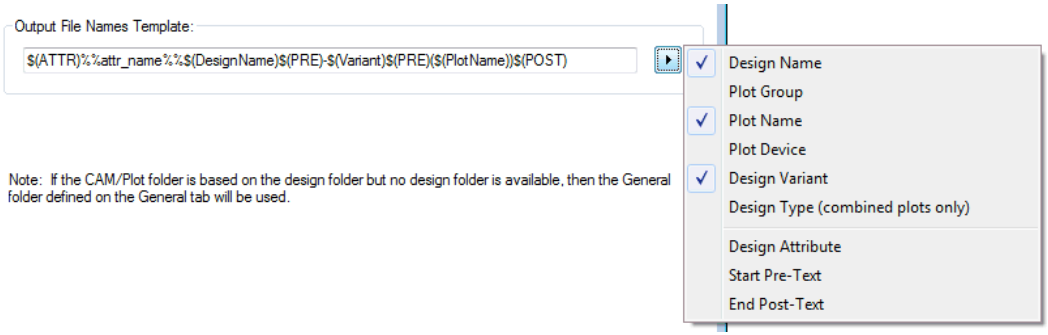
The **Folders** dialog **CAM Plot** page also has the same new controls.

Most keywords are straightforward, but there are a few that need additional explanation:

$(PRE) and $(POST) are used to add pre-text and post-text either side of a variable field. For example "$(PRE)[$(Variant)]$(POST)" will be replaced by "[USA]" for a plot for variant "USA" and will produce nothing for a plot with no variant.

$(DesignType) is replaced by "pcb" for a pcb design and "sch" for a schematic design.

$(CombinedType) is the same as $(DesignType) but is only used if the filename is a combined output containing several plots (like pdf).



The **Design Attribute** keyword can be used to insert the value of design level attributes into the CAM/Plot and Report file names. This is available in the **Reports** dialog, **Cam/Plot Settings** dialog and the **Folders** dialog.

Choosing this keyword from the menu will add **$(ATTR)%%attr_name%%** to the template, where **attr_name** must be replaced in the template with the name of the attribute you want to use, e.g. If the design has an attribute **JOB_NO** with the value **1806**, then **$(ATTR)%%JOB_NO%%** will be replaced with **1806** in the name.

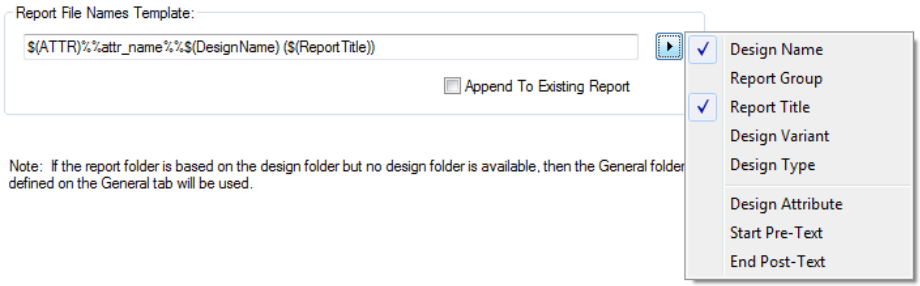
Reports

Similar system to CAM Plots. Rather than using a drop down list to define when it included in the report filename, you can now design the filename and folder using keywords in a name template.

In the **Generate Reports** dialog in **Reports**, the **Report Names Based On** section had a drop down list which has been now replaced with a **Report File Names Template** section with a template edit box and a popup menu button to make it easier to add the keywords. There is also a popup menu button next to the folder edit box as well.

The **Folders** dialog and **Reports** page also has the same new controls.

The "$(Report Group)" keyword will be replaced with the new report group the report is in. On the **General Reports** page there is a **Report Group** box to supply the group for each of the standard reports. **User reports** can have a report group assigned by including the new **Set Report Group** command in their script.



Adding Conditional Report and CAM Plot File Templates

There is now a set of advanced keywords that can be used to create conditional **CAM Plot** and **Report folder** and **filename** templates. For example, this functionality can be used to create completely different plot file names for some of the different plot types. This effects the **Reports** dialog, **CAM Plot Settings** dialog and the **Folders** dialog.

There are a set of advanced keywords that can be used to create conditional folder and filename templates. For example to have completely different report file names or folders for each of the different design types. The following keywords are not on the menu as they have to be combined as a set, **but they can be inserted by hand**. Care must be taken when inserting conditional templates and should only be attempted if you know what you're doing.

$(IF) $(MATCHES) $(THEN) $(ELSE) $(ENDIF)

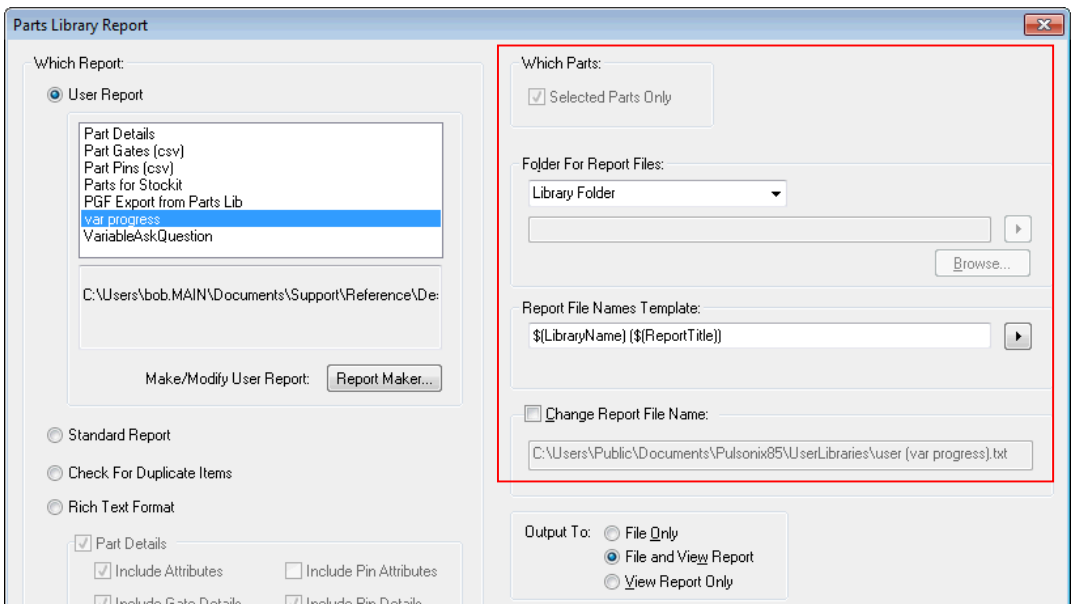
If the text following the "$(IF)" keyword matches the text following the "$(MATCHES)" keyword, the text following the "$(THEN)" keyword will be included in the name, otherwise the text following the "$(ELSE)" keyword will be included. There must always be a "$(ENDIF)" keyword at the end of the condition. Using "$(MATCHES) allows you to use wildcard characters in the text following it, but you can replace it with keywords "$(=)" or "$(!=)" to test exactly equals or not equals respectively. Note: nested conditions are not allowed, i.e. an "$(IF)" keyword should not be placed inside another "$(IF) ... $(ENDIF)" sequence.

For example, for design "myDesign.pcb" with design level attribute "Issue" with value "V02" the following template "$(DesignName)-$(IF)$(DesignType)$(MATCHES)PCB$(THEN)$(ATTR)%%Issue%%$(ELSE)Drawing$(ENDIF)" will produce "mydesign-V02" for a PCB design and "myDesign-Drawing" for a schematic design.

Specifying Parts and Symbol Library Report Names and Locations

The formatting and location of reports from within the **Library Manager** for both the **Schematic Symbols** and **Parts** has been modified to enable more accurate reporting.

The dialog below shows the **Parts Library Report** page:



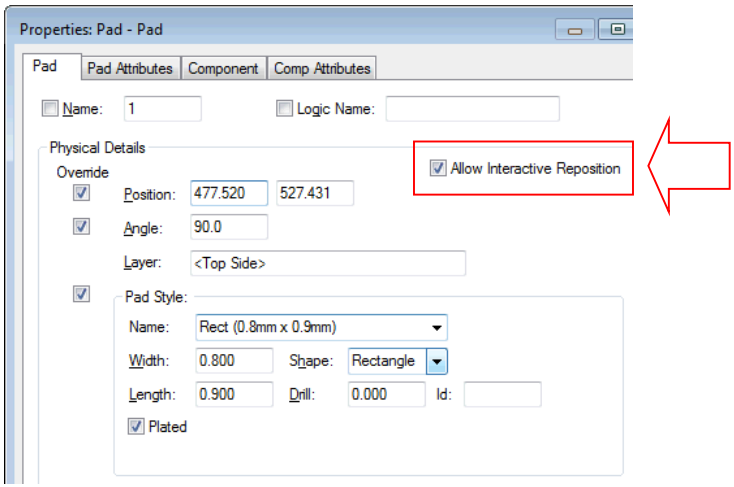
The **Report File Name Template** button opens an additional menu from which to make name selections:

Override Component Pad Position

As well as the **Style**, you can now override a **Component Pad position** and **rotation**. This is a generic feature and so also applies in a Schematic as well as a PCB. It also applies to **Doc Symbol Pads**, **Component Mounting Holes** and **Component Vias**.

This is presented in the **Properties** dialog for a **Component Pad** as a set of **Override** check boxes. These replace the previous **Alternative Pad Style** check box and the **Override** switch for pin type.

Note. The **Layer** cannot be overridden and is displayed here for information only.

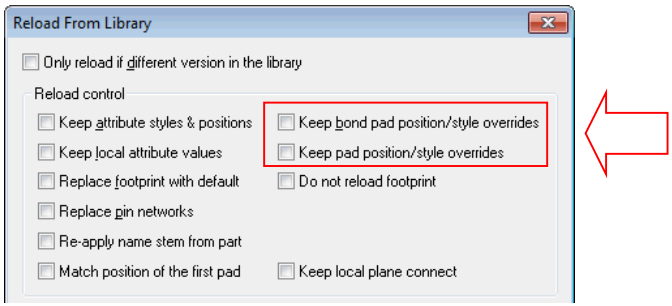


There is also a new **Allow Interactive Reposition** check box. If selected, the pad can be interactively **moved** and **rotated** to reposition it. This also allows you to align it with another item. By default this is unchecked to not allow interactive repositioning.

Reload From Library

Reload From Library now includes additional switches **Keep bond pad position/style overrides** and **Keep pad position/style overrides**, which replace the **Keep bond pad positions** and **Keep alternative pad styles** switches.

There is also now a **Keep pin positions/style overrides** switch in Schematics.



Design Revision Analysis

The **Design Revision Analyser** option will now report all **overridden pads**.

Report Maker

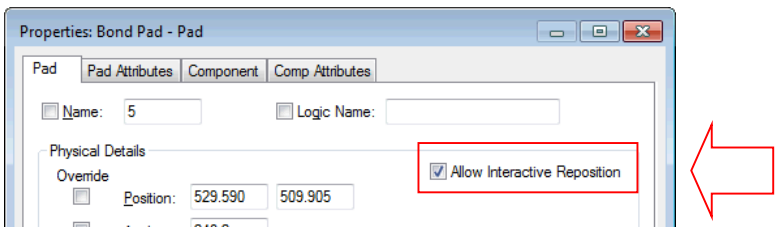
The **Report Maker** option now has a set of **Is XXXX Overridden** commands and a set of **Original XXXX** commands. For example **Is Position Overridden** and **Original Position**.

Note. **Is Style Overridden** replaces **Is Alternate Style** and **Is Pin Type Overridden** replaces **Has Local Pin Type** commands. These commands can be used to report where component pads have been physically changed in a design.

There is a new format file **Overridden Pad Details** that is installed by default with Version 8.5

Allow Interactive Reposition for Bond Pads (Chip Packaging Toolkit Option)

If you have the Chip Packaging Toolkit option, Bond pads will now use the new **Allow Interactive Reposition** feature to enable you to lock them against repositioning. When a component with bond pads is added to the design, all bond pads will initially have **Allow Interactive Reposition** off, and so they cannot be accidentally moved. By default this is unchecked to not allow interactive repositioning.



Copy Tags status in Attribute Editor

The **Copy Tags** option in the **Attribute Editor** now also applies to **pasting**. When pasting with **Copy Tags** off, the visibility is kept as it was.

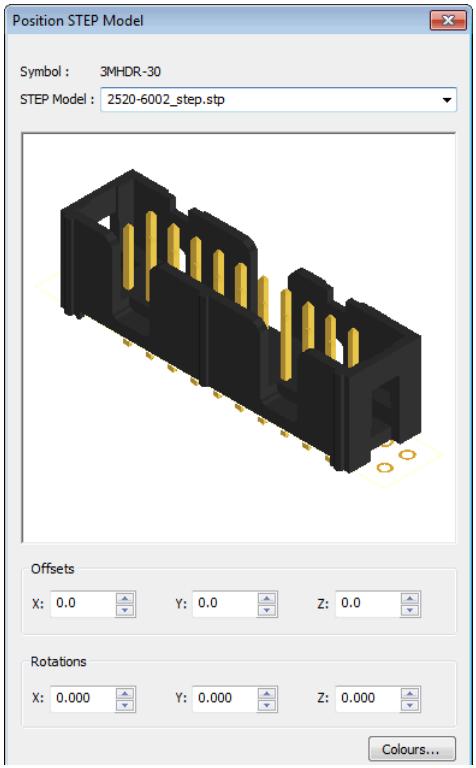
*As a reminder, the **Copy Tags** option controls the copying and pasting of the 'visible', 'fixed' and 'undefined' state information. If you want to copy just the text without the state tags you can uncheck the **Copy Tags** option.*

Position STEP Model

Within the Footprint Editor and PCB Doc Symbol Editor, a graphical method of defining <STEP Filename> <STEP Offsets> and <STEP Rotations> attributes has been added.

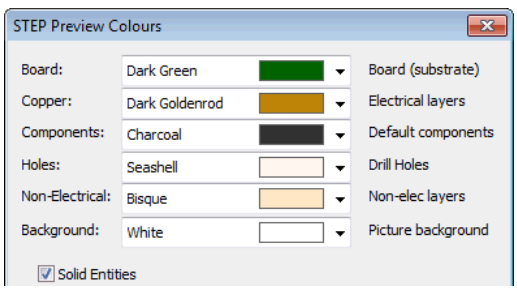
The **Position STEP Model** option is available on the **Tools** menu.

Selection of the **STEP Model** name is by way of the drop down list box from models available in your system.



Offset values and **Rotations** are added by typing them into the appropriate boxes. Once values have been added, the preview of the model on the footprint will be updated. This allows you full graphical control and visualisation of the changes that you make.

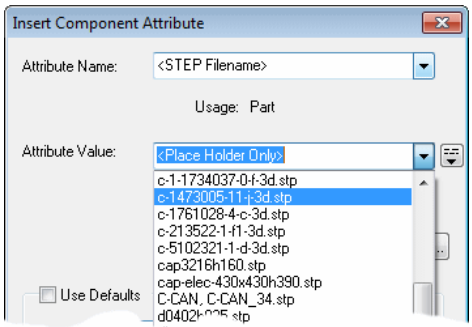
Colours - use the **Colours** button to set the colours and solid entity status of the Preview window.



Select STEP Filename Model from drop down list

When using the **Insert Attribute** option, for the **<Step Filename>** system attribute, you can now use the drop down list to select the STEP model filename. This list is read from the STEP model names in the search list.

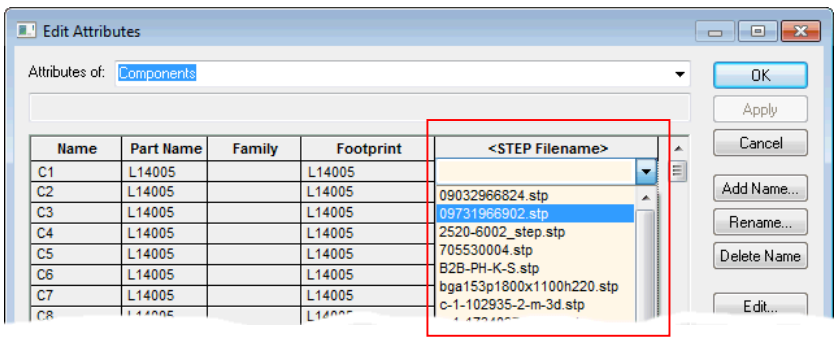
This is available when adding the **<Step Filename>** attribute in the design using **Properties**, through the **Attribute Editor**, or when adding this attribute in the **Footprint Editor** and in the **Part Editor** when adding this attribute.



On selection, you are presented with a drop down list containing <Place Holder Only> plus a list of any available STEP model filenames.

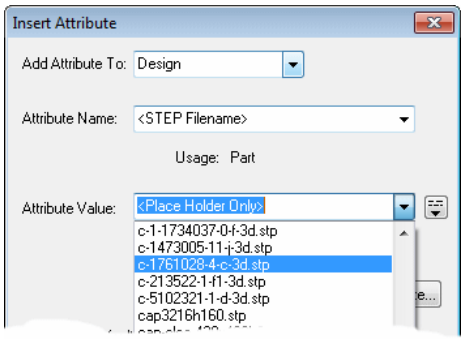
STEP Model Name in Attribute Editor

When using the Attribute Editor, if you choose <STEP Filename as a new attribute name, you can then choose the STEP model name from the drop down list presented within the editor cell.



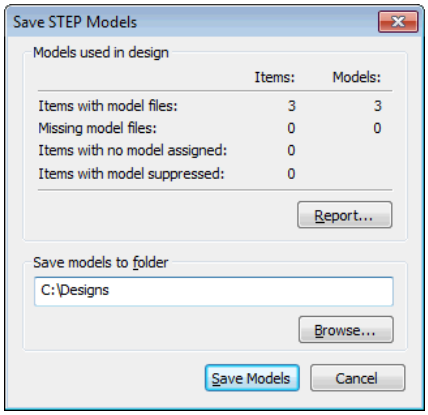
Attribute <STEP Filename> available for Doc Symbols

The **<STEP Filename> attribute** can now be used in **PCB Doc Symbols**. It works the same way as for Component footprints. This can be used for items such as mounting screws and mounting hardware.



Save STEP Models Option

There is a new command for a PCB design to save a copy of all the step models used in the design to a separate folder. The **Save STEP Models** command is available on the **Utilities** menu.



This, along with the changes to specify separate folders for important CAM files and reports, makes it easier to create a release folder tree containing all important files that can easily be archived or sent for review etc.

The dialog will show you the results of analysing the components and documentation symbols in your design for STEP model file usage.

- How many have model file names assigned where those model files exist in your STEP model folders, and the number of model files found.
- How many have model file names assigned but those model files cannot be found.
- How many have no model file name assigned.

- How many have a model file 'suppressed' (by adding the STEP Filename attribute and leaving the value blank).

You can click the **Report** button to get a simple report of the models found (and not found) for your design.

To save the model files to a particular folder, use the **Browse** button to locate the desired folder, then click **Save Models** to make a copy of all model files found, from their existing location into that folder.

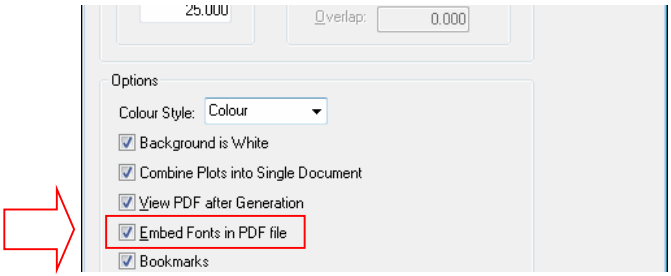
Report

A sample report provides you with this information:

```
STEP Models
-----------
Report Written : 25/11/2014 12:07:15 PM
Design Path    : C:\Reference\Designs\STEP Example.pcb
Design Title   :
Created        : 01/02/2011 11:31:26 AM
Last Saved     : 12/10/2012 11:17:25 AM
Editing Time   : 17 min
MODELS FOUND FOR ITEMS IN DESIGN
--------------------------------
    Model name                      Items using model
    ------------------------------------------------
    c-5102321-1-d-3d.stp            1
    dpak3p-841x1018h444.stp         1
    plcc-44p1658x1658ls127h439.stp  1
MODELS MISSING FOR ITEMS IN DESIGN
----------------------------------
    None
MODEL FOLDERS
-------------
    Model                           Folder
    ------------------------------------
    c-5102321-1-d-3d.stp            C:\Reference\STEP Models
    dpak3p-841x1018h444.stp         C:\Reference\STEP Models
    plcc-44p1658x1658ls127h439.stp  C:\Reference\STEP Models
```

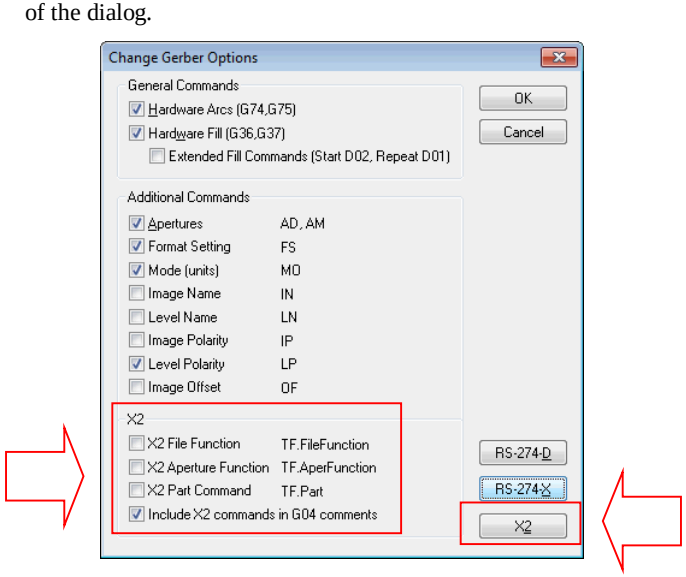
PDF Output – Embed Fonts

When using the **CAM Plots** output and **PDF**, you can now choose whether you want to embed fonts in your PDF. In the PDF device setup dialog, select the **Embed Fonts in PDF file** to embed the font, previously this was done by default. For example, Arial Unicode is a very large font on disk and hence the output PDF is also very large.



Support for Gerber X2

From within the CAM Plots dialog you can now select X2 as the Gerber format for export. This switch is in addition to the existing RS-274-D and RS-274-X formats. By default, this option will be pre-selected with the RS-274-X format. Selecting X2 will enable the three new options at the bottom of the dialog.



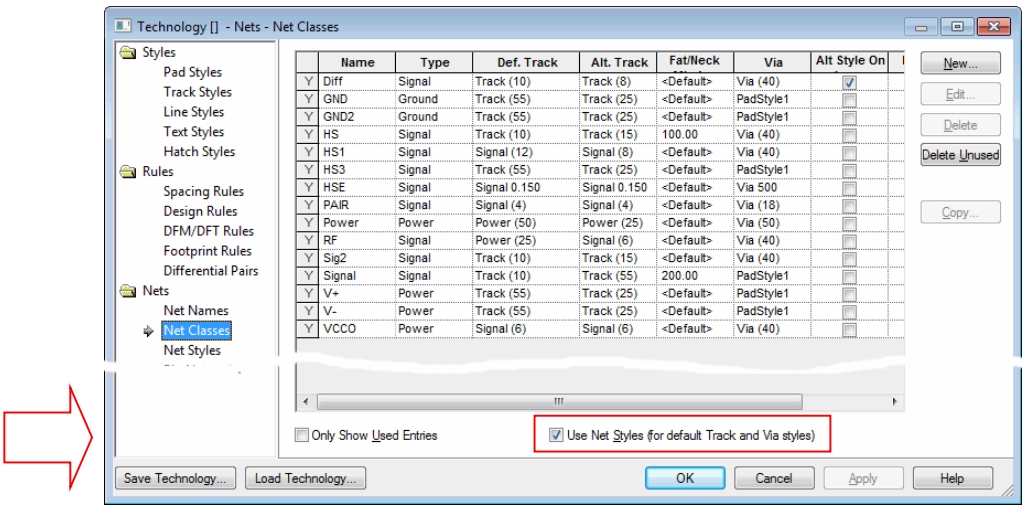
At any point, individual check boxes can be selected to enable elements within the Gerber export format. Note that some of the commands for X2 may appear in the 274-X format as well.

Cross Probe of Star Points

You can now cross probe **Star Points** in your designs. Note: Star Points must be defined as Pulsonix Star Points using the special Doc Symbol type.

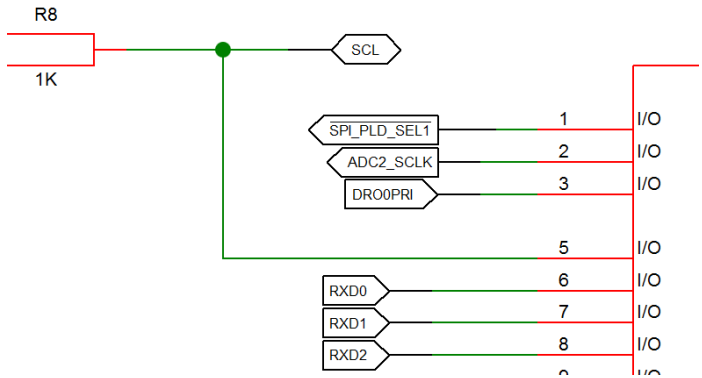
Enable Net Styles From Technology Dialog

As well as enabling the **Technology Net Styles** from the **Design Settings** dialog, **General** page, you can now also enable it from with the **Technology** dialog and **Net Class** page as well. The same verification checks and warning dialog will be displayed; this is just a more convenient method to enable it.

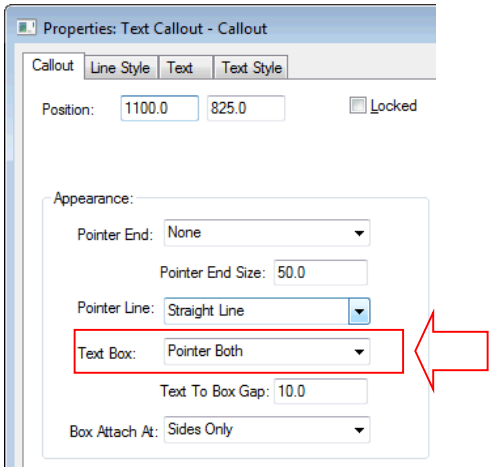


Text Callouts with Net Names

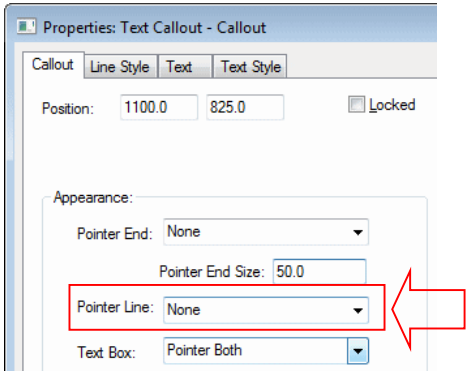
Text Callouts can now be attached as part of a single pin Doc Symbol to define a signal reference symbol, A Net Name attribute can also be attached within the Text Callout which, when used in the design, will dynamically resize to fit the net name length.



Three new **Text Box** shapes have been added to a callout definition: **Pointer Left, Pointer Right and Pointer Both**. All three examples are shown in the picture above.

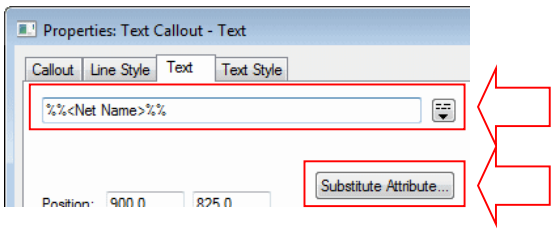


You can also make the **Pointer Line: None** for the connecting line between the box shape and the terminal if you just wish to display the box shape.



Text Callouts and Net Name Attribute

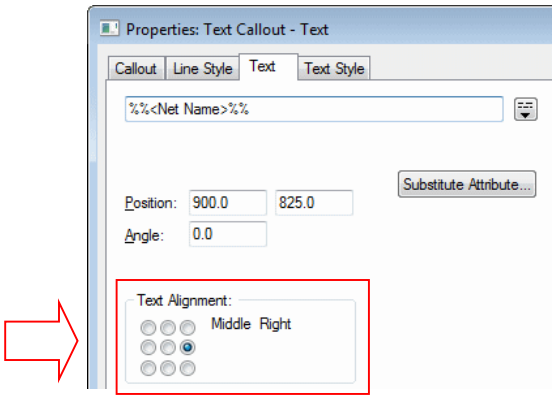
When adding a **Net Name attribute** to the **Text Callout**, use the **Attribute Substitution** button to add the system %%<Net Name>%% attribute. The %% is required if manually typing in the attribute name so that Pulsonix knows to pick up the attached Net Name and not just simply display plain text.



Text Callouts and Net Name Text Alignment

Within the **Text Callout** box, you should also be aware of how the actual **Net Name attribute name** is positioned relative to the box itself. It is advised that you position it using one of the 'Middle' **Text**

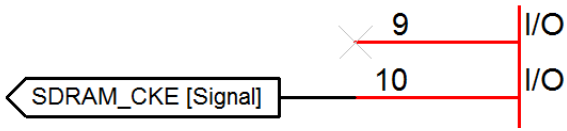
Alignment options and either **Left**, **Right** or **Centre** depending on the Pointer direction. This way, the box will 'grow' dynamically in a suitable direction depending on the Net Name size.



Text Callouts in the Design

You can now attach **Text Callouts** to **Component pins** in a design to display information about the pin. Previously, it always attached to the whole Component. Again, this can be used instead of displaying a Net Name on a pin if you want a box around the net name. This can be an alternative method to using the new Doc Symbol Text Callout and Net Name mechanism although you will have to edit the Text Callout Properties in order to get the required parameters.

The Attribute Substitution in Text Callouts can be used to create some useful combinations such as Net Name and Net Class, for example %%<Net Name>%% [%%<Net Class Name>%%] (using additional square brackets for notation).

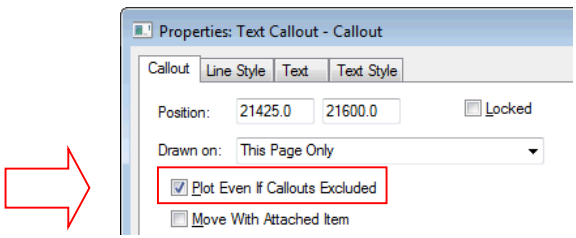


Text Callouts and Schematic Junctions

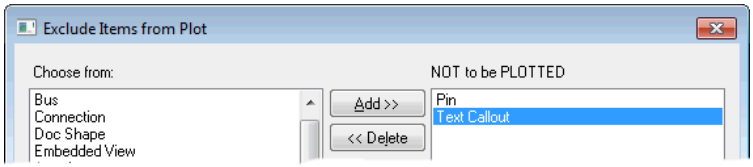
You can now also attach **Text Callouts** to Schematic **Junctions** and **Bus Terminals**.

Override Exclude Callouts From Plot

There is a new check box **Plot Even If Callouts Excluded** on the **Callout Properties** dialog for Schematic and Schematic Symbol designs. Use this if you are generally excluding Callouts from Schematics plots, but require some Callouts to be plotted, for example Callouts that are used to display net names.



Below shows the **CAM Plot dialog** and the **Exclude** dialog. With this situation and **Text Callouts** generally excluded, it the **Plot Even If Callouts Excluded** check box is selected, then specific **Callouts** will be plotted.

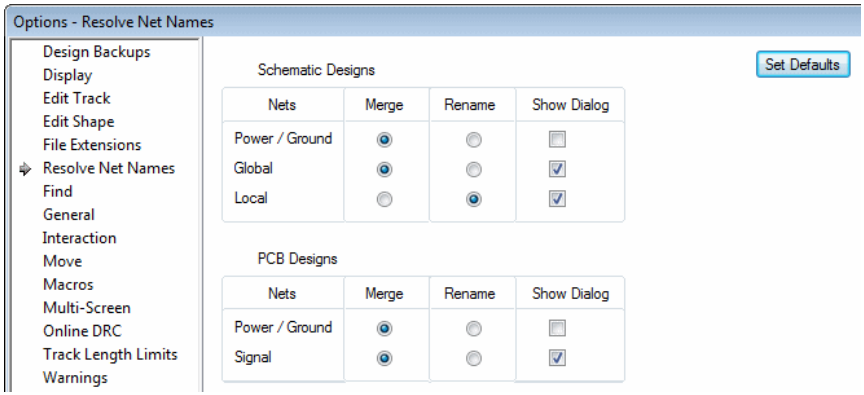


Preset default Resolve Net Name parameters

The default action of the **Resolve Net Names** dialog is now determined by options set up on the new **Options** dialog, **Resolve Net Names** tab.

You can choose to default **Merge** or **Rename** on **Power/Ground**, **Global** and **Local** nets. You can also choose if the **Revolve Net Names** dialog is shown when your selection contains nets of certain types.

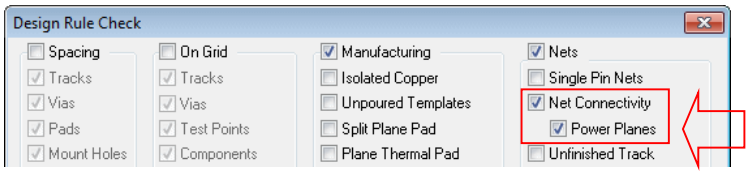
These choices can be made differently between Schematic and PCB designs.



Additional DRC Checks

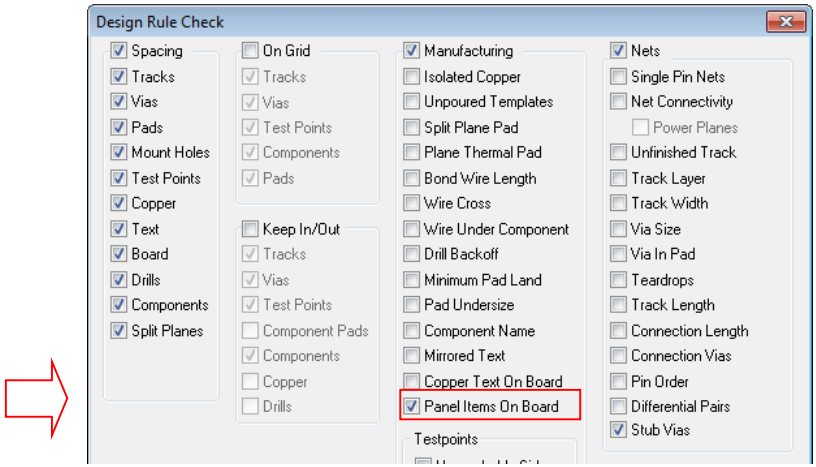
Net Connectivity Check

The **Net Connectivity** check now has a **Power Plane** option which does a more in depth check of breaks in power plane copper - in particular split planes. Note: This can be much slower than the normal Connectivity check.



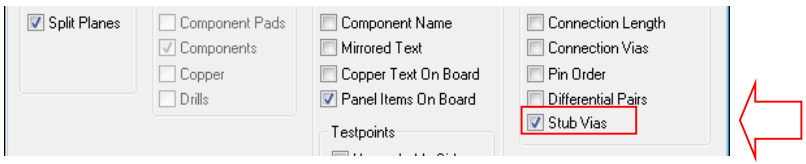
Panel Item Check

There is a new **Panel Items On Board** check for use when creating panels. This new check will be used for items which have been flagged on their Layer Class using the new **Check Items Outside Board** check. See below under ***Panel Checks in Layer Class dialog.***



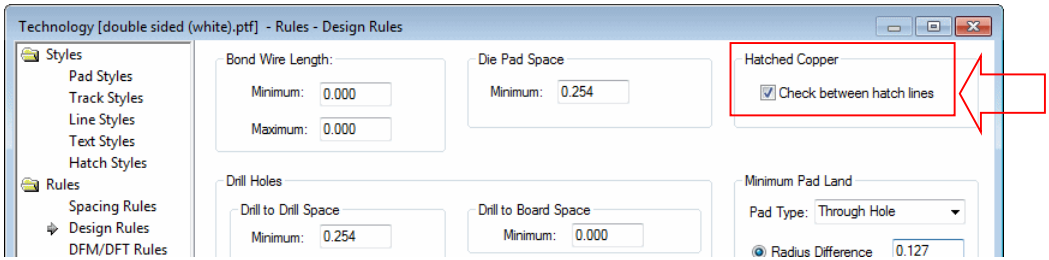
Stub Vias Checks

There is a new **Stub Vias** check in the DRC dialog under **Nets**. This will check for tracks with dangling ends, or ending on vias which do not appear to connect to template or plane.



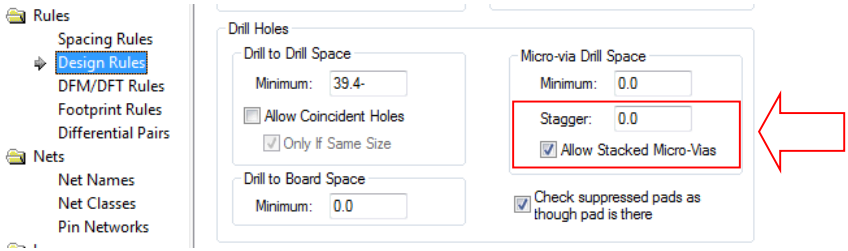
Checking of Cross Hatched Copper

With the **Hatched Copper** check option in the **Technology** dialog and **Design Rules** tab enabled, the **DRC** and **Copper Pour** options now take the gaps between hatch lines into account when reporting spacing errors, and when avoiding obstacles when pouring.

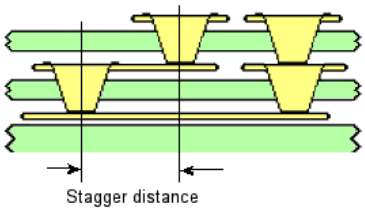


Staggered Micro-Via spacing

In the **Technology** on the **Design Rules** page, you can specify a minimum stagger distance between micro-vias on adjacent layer spans. There is also a check box (which defaults to on), to allow stacked micro-vias even when a stagger distance is specified.



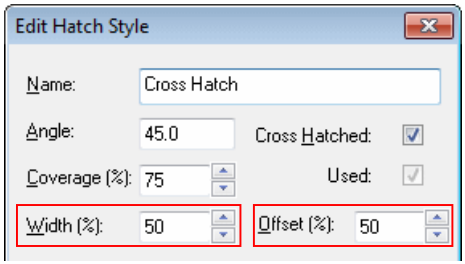
The **Stagger** distance, is the minimum centre to centre distance between micro-vias on adjacent layer spans.



Allow Stacked Micro-Vias allows centre aligned micro-vias on adjacent layer spans, even when the Stagger distance is not 0. You can automatically insert Stacked Micro-Vias using Composite Spans.

Hatch Lines Using Percentage Width and Offset

From within the **Technology** dialog and **Hatch Styles**, you can now specify the width of the Hatch Lines as a percentage of the line style. You can also offset a hatch style as a percentage of the separation between lines. These controls have been added to provide you with more control over the poured hatched copper.

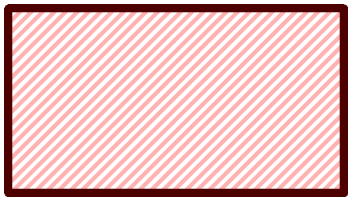


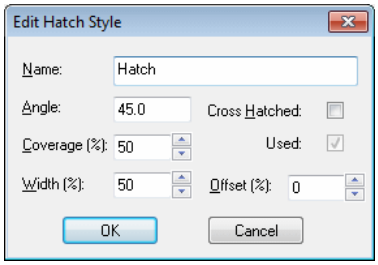
Width Percent gives a hatch line width in terms of percentage against the Template shape line style width. This is useful if you wish to adjust the hatch line styles without adjusting the Template outline. Must be greater than 0 and less than or equal to 100%. The default setting is 100%

Offset Percent gives the offset of the hatch lines from its original position, given as a percentage of the line separation. You must wish to 'move' (offset) the hatch line positions to 'hit' or avoid other obstacles within the pour. The value must be between 0 and 100%. The default setting is 0%

Below are various scenarios to illustrate the affect the two new settings have on poured copper:

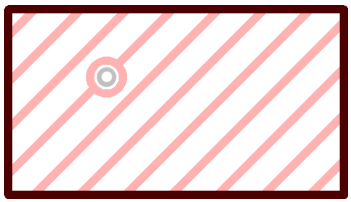
'Normal' poured copper showing the hatched line style being the same width as the template.

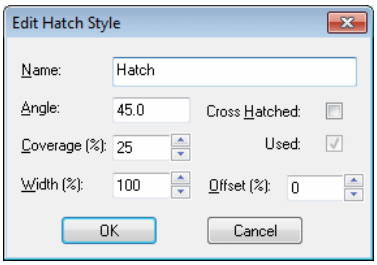




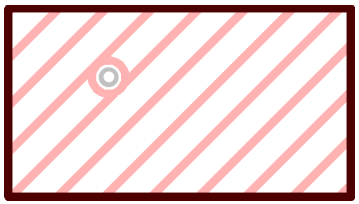
Using a Width: adjustment of 50%, the hatch line style is now 50% thinner than the template width.

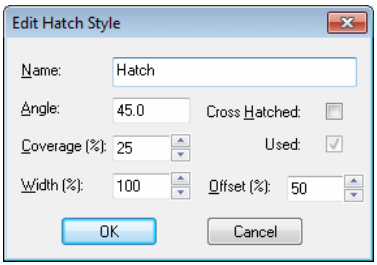
This picture is zoomed to show the effect.





This illustration shows an offset of 0% (normal); the hatch line is centred on the pad.



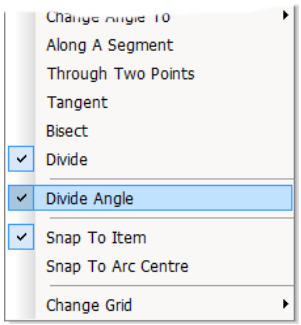


This shows the effect of making an Offset of 50% to its original position. The pad is now straddled between two hatch lines.

Construction Lines

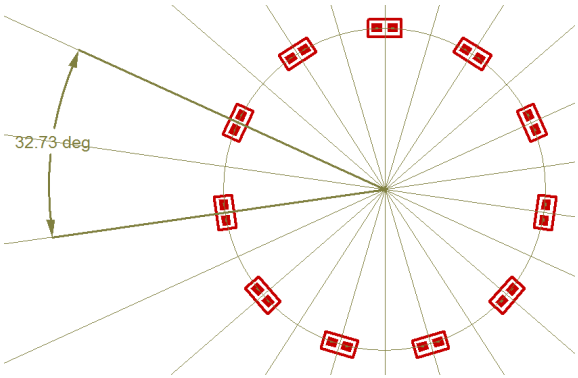
Dividing a Circle

You can now can use the **Divide** mode with **Divide Angle** enabled to add lines to divide up a circle (or circular pad). The context menu option is now called **Divide Angle** (previously, it was **Divide Angle Between Two Lines**).



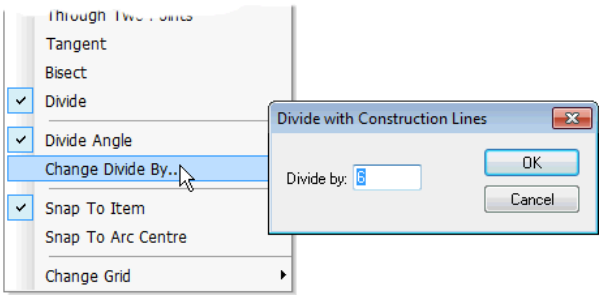
When in **Divide** mode and are dividing angles, you now just click on an arc or circle to divide it (previously you had to Ctrl-click to select arcs). A prompt on the status bar now shows that you can pick arcs and circles.

Using the Divide function within Construction Lines to divide a circle allows you an alternative to using Polar Grids (which can still be used). If your requirement is calculated to an uneven number of lines, say 11, the Divide number must be doubled on addition. This is because Construction Lines are infinite lines and do not stop at the centre of the circle. This is shown in the example below where 11 LEDS are placed on equal increments around the circle.

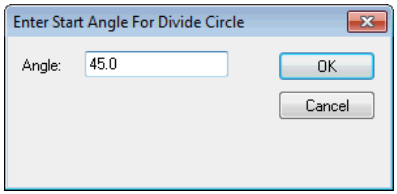


Change Divide By

When in **Divide** mode, there is now a new **Change Divide By** option on the shortcut menu to change the divide number. This can be used if say, you first choose to divide by 2, then subsequently need to divide by 3.

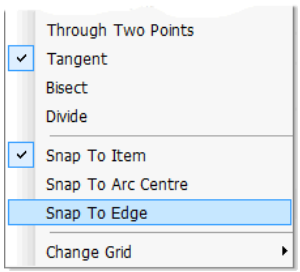


Also, if dividing an angle and you select a circle or round pad you will be presented with a dialog to specify the start angle for the dividing lines.



Snap to Edge

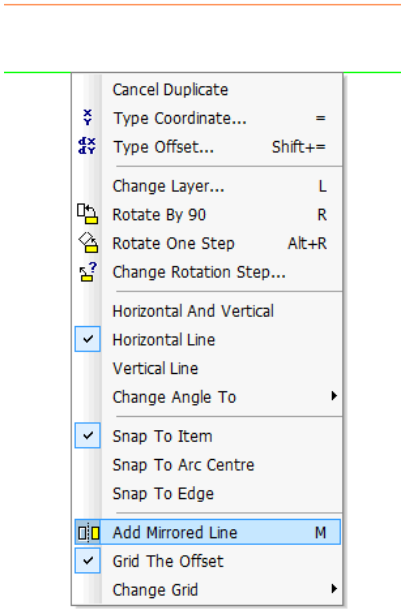
A new option, **Snap To Edge**, has been added. When selected from the context menu, this will cause the construction line being added to snap to the nearest edge of the picked item, rather than snapping to its origin or centre line if it is a segment.



Add Mirrored Line

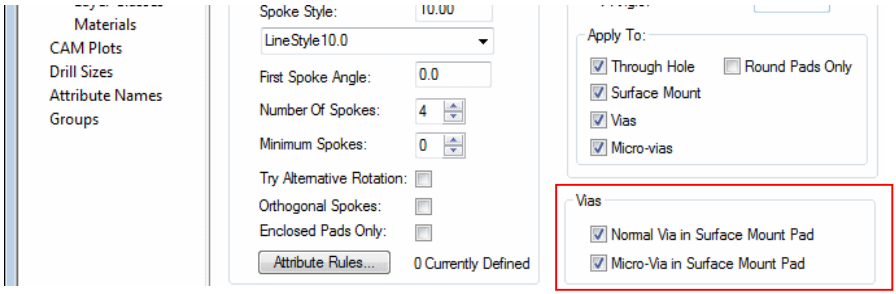
When adding a single construction line there is a new option **Add Mirrored Line** available on the context menu if you had a pre-selected construction line, or have used Ctrl-Click on an existing construction line, or are using duplicate.

Use this to add a copy of the current dynamic line, mirrored about the original picked line.



Via In Surface Mount Pads

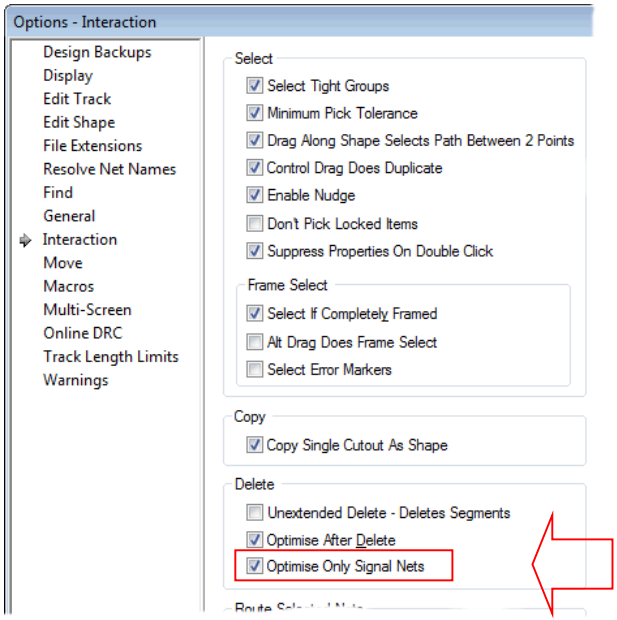
From within the **Technology** dialog and **DFM/DFT** page, you can now allow vias in surface mount pads. You can also control **Normal** and **Micro-Vias** separately.



These are used when running the **Via In Pad** check in the **DRC** option.

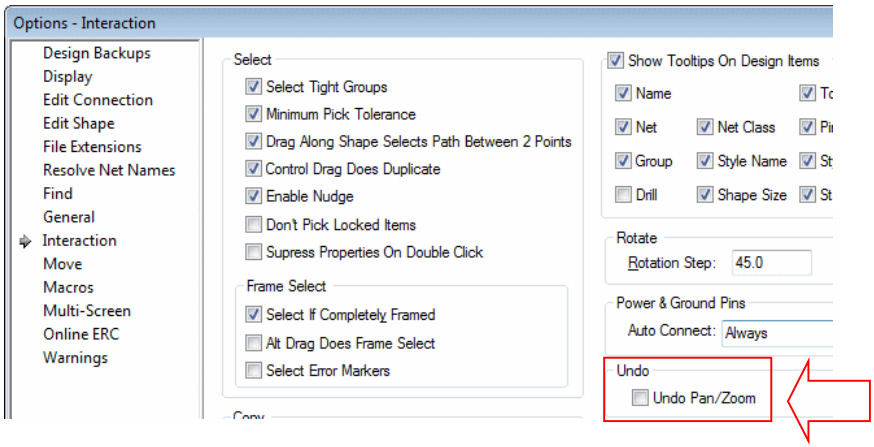
Optimise on Delete

There is a new option to only perform optimise on signal nets when performing delete and other related operations, such as unroute track path. This significantly speeds up performance on larger designs, whilst still optimising most nets. The **Optimise Only Signal Nets** option is located on the **Options** dialog and **Interaction** page.



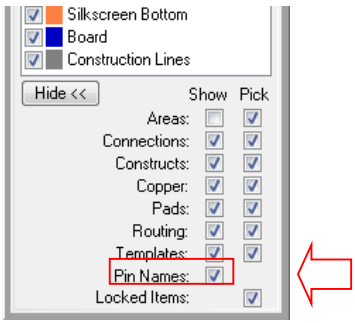
No Undo/Redo on Pan/Zoom

From within the **Options** dialog and **Interaction (PCB & SCM)** tab, there is now an option to turn off **Pan/Zoom** for **Undo**. With this switch unselected, if an Undo is performed, it means that only the function will be undone and not the Pan or Zoom levels. Select this option to perform Undo as it previously was.



Layers Bar - Show/Hide Pin Names

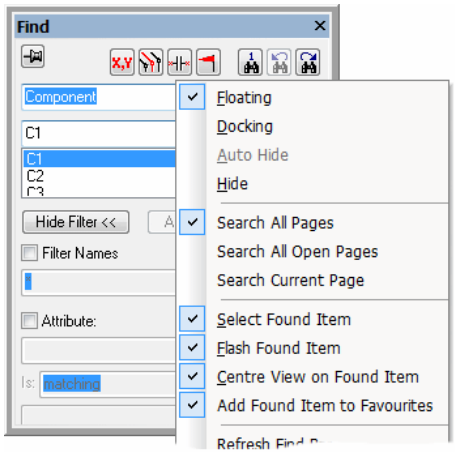
The **Show/Hide Pins Names** option is now also available on **Layers Browser Bar** in the PCB design editor. You can still also use the **Colours** dialog or **Component Properties** dialog to change their display status.



Find Changes

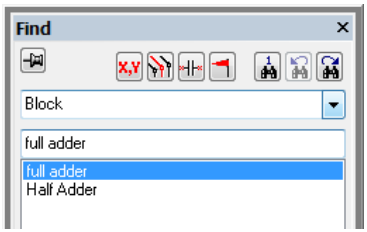
Option to Find in open pages only

Find now has an option to **Search All Open Pages**. This is available on the context menu when using the Schematic editor.

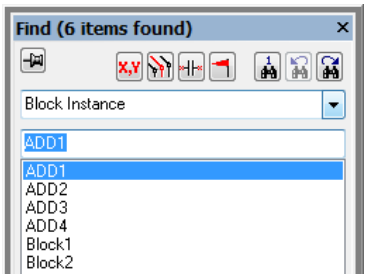


Find by Block Instance

You can now **Find Block Instances** by block.



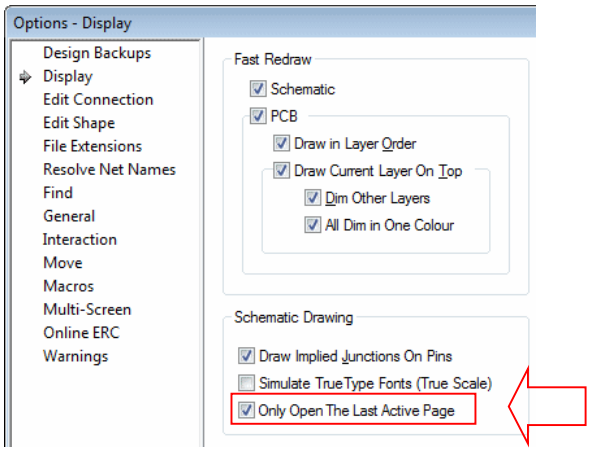
The above example shows Find on a **Block** where two different Block types have been added (by adding the Half Adder example).



The above Find example is of the same Half Adder block added twice to the design. The **Block Instances** are broken down to individual instances so they can be located by instance name.

Option to Only Open Last Active Page

There is a new option in the **Options** dialog and **Display** page to **Only Open The Last Active Page** in a Schematic design. This saves time when opening a large design with many pages open; it will only open the last active page that you had open at the time of closing.

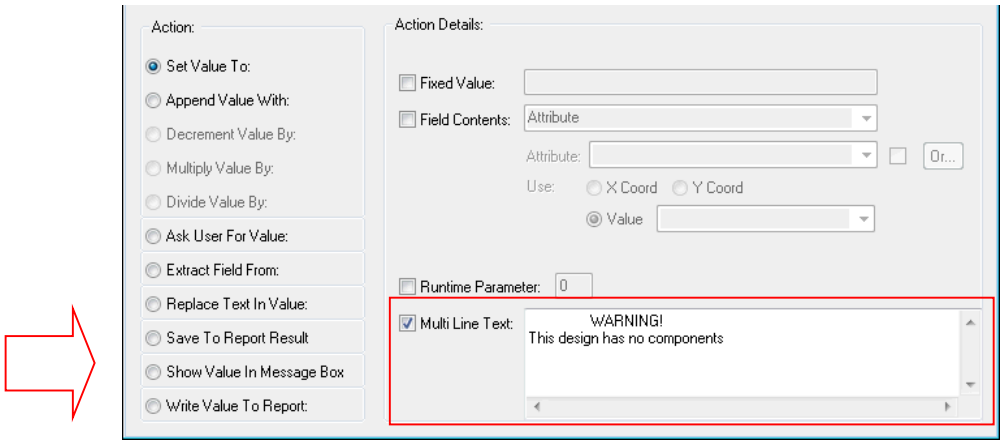


Report Maker – New Commands

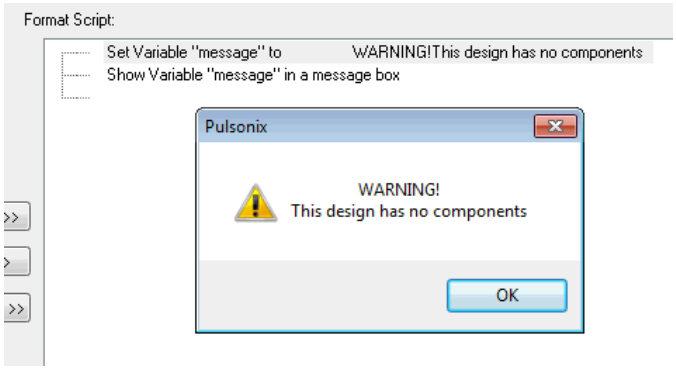
Two changes have been made to the Report Maker:

A new command has been added – **Is Via In Sm Pad** This will test if a drill hole is a via in a surface mount pad.

The **Variable** command has been modified. You can now initialise a variable with multi line text.



When the script is run, the message displayed will be multi-line.



Using Reports Folder Template Keywords in Report Maker

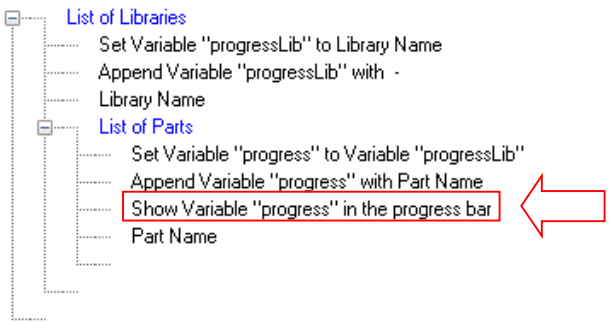
You can now use the new **Reports folder** and **file name template** keywords within the **Report File Name** command in the **Report Maker**. *See the Adding Conditional Report and CAM Plot File Templates section above.*

Changes to the Variable Command

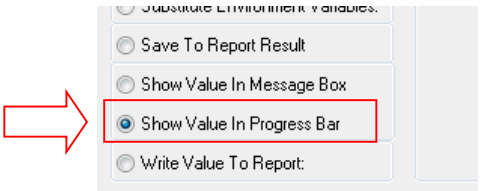
The **Variable** command has been enhanced:

Changing Text in the Progress Bar

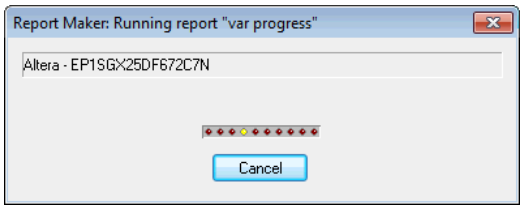
You can use a variable to change the text in the progress bar, this is useful when running library-based reports for example.



The **Edit Variable** dialog will allow you to select **Show Value In Progress Bar**.

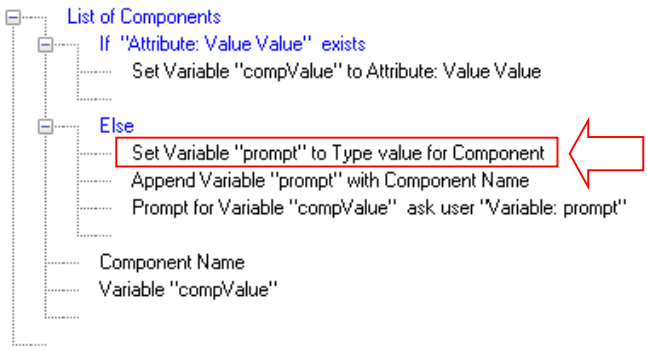


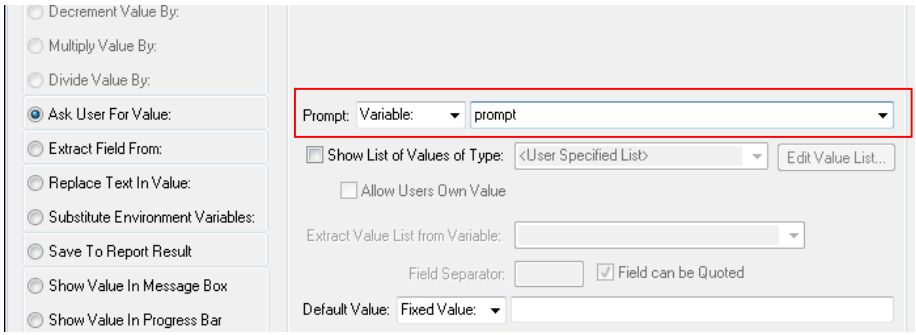
When run, the progress dialog (for this example) will insert the Library name to show which one it is accessing.



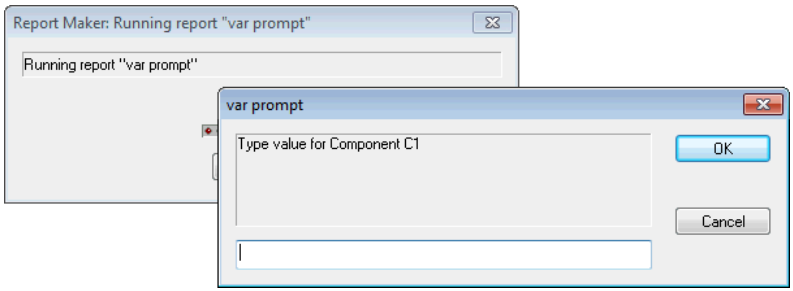
Prompt String in a Variable

When using a variable prompt command, you can now use a variable to provide the Prompt string and default answer at runtime. For example to prompt “Add value for Component R1”.



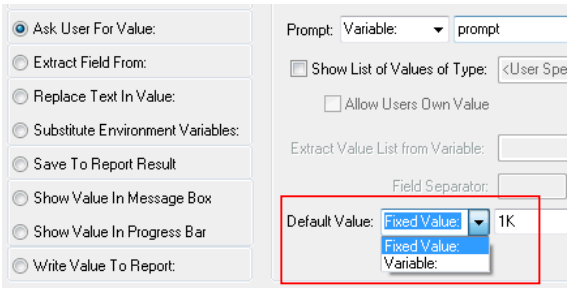


When run, a prompt message is presented:



Prompt String in a Variable – Default Value

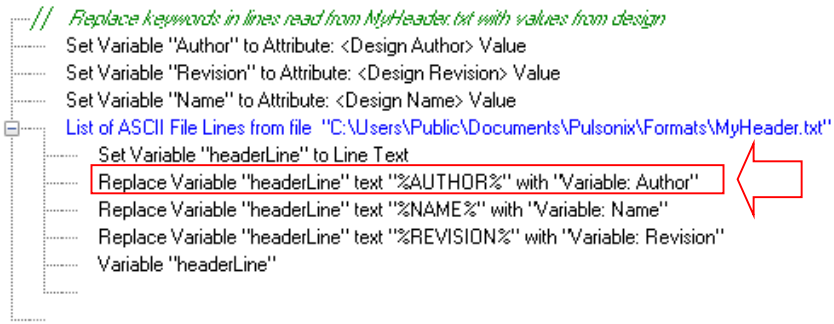
Using a variable to provide the Prompt string, you can also set the **Default Value** for the prompt.



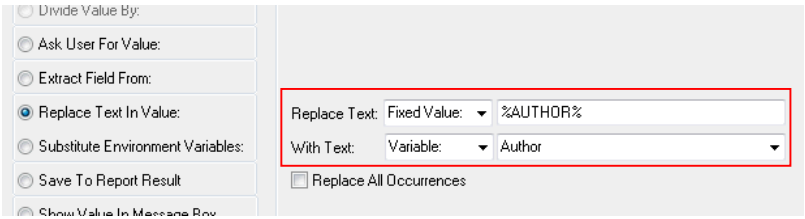
Once run, this default value will appear in the prompt dialog but can be overwritten with a typed value. The drop down list will also allow you to set the default value between a **Fixed Value** which you type, or a **Variable,** selectable from the drop down list of defined variables.

Variables within the Replace Text Command

For the **variable replace text** command, you can now use variables to provide the replace text and replace with text at runtime. In the example below, the replace text is extracted from the text file.



The **Variable** dialog now shows the **Replace Text** value and **With Text** as a drop down list boxes.

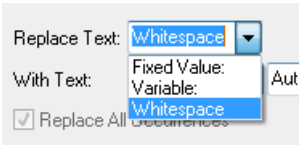


Contents of C:\Users\Public\Documents\Pulsonix\Formats\MyHeader.txt

```
============================
Design Name: %NAME%
Designed By: %AUTHOR%
REVISION:    %REVISION%
============================
```

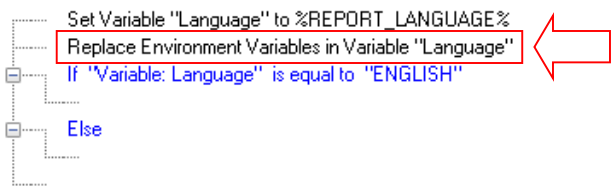
Whitespace characters

Also changed: when replacing whitespace characters is now in the drop list, rather than being a separate check box.

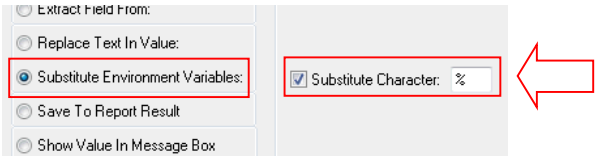


Substituting System Environment Variables into the Variable Value.

There is now a new variable command to substitute system environment variables into the variable value.



The **Edit Variable** dialog will allow the **Substitute Environment Variables:** check box to be selected



Selecting the **Substitute Character:** check box allows you to define the **Substitute Character** if it is going to be different to your default. If you leave it unchecked, the default will be taken from the **Options** dialog and **General** setting.

Minor Scripting Changes

Scripting in V8.5 has now been extended to facilitate **Library Reporting**.

A new **RunLibraryReport** method has been added for an **Application** object, and a new **RunReport** method has been added for a **PartLibrary** object. The Online help for Scripting has more specific details on how to use this implementation.

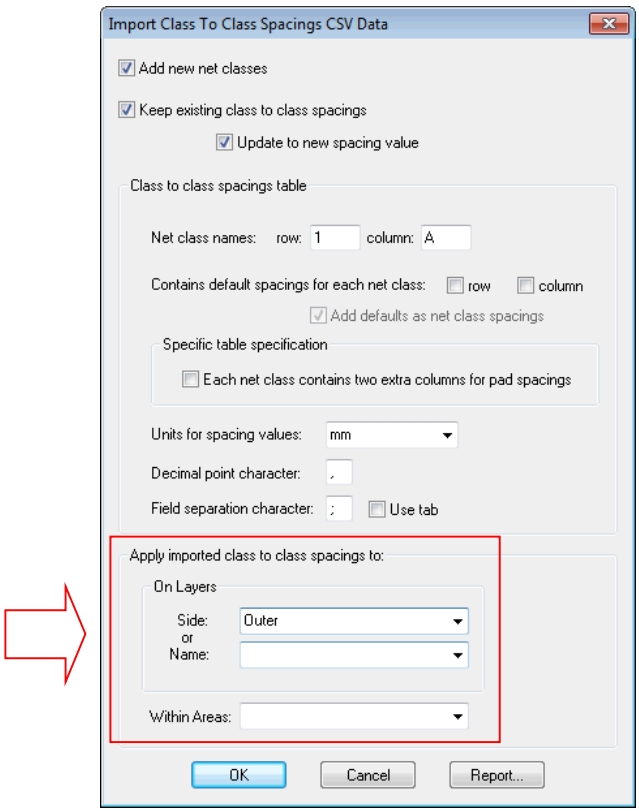
By providing a script file on the command-line, it can be used within a scheduled task and when launched in conjunction with the /hidden switch, it can be used with the Task Scheduler 'Run whether user is logged on or not' option set.

Enhancements to Import Class To Class Spacings CSV

Spacing CSV Applied To Layers and Areas

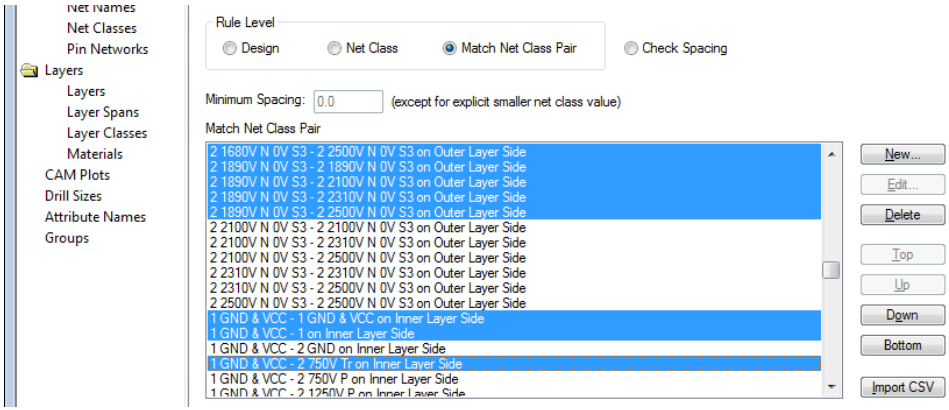
When importing **Spacing CSV** files through the **Technology** dialog, **Spacings**, **Match Net Class Pair Import** dialog, you can now choose which **Side**, **Layer** and **Area** to use when importing a CSV file table of class to class spacings.

There are extra controls for this at the bottom of the **Import Class To Class Spacings CSV Data** dialog. Set **Side** to **<Any>** and leave **Layer** and **Area** blank to import spacings for all layers.



Multiple selection

You can now select more than one class pair in the list (using standard Ctrl, Shift and combination keys). The **Delete** and **Move** buttons will act on the multiple selected items. If the selected items are not sequential, after using one of the move buttons they will become sequential. Newly imported Class to Class Spacings will be added to the top of the list (as before) and will now be selected. This enables you to move them all to the bottom of the list by pressing the **Bottom** button for example.



Floating Network Licensing – Disconnect Client at Admin Level

If you are using a floating network license system for your Pulsonix licensing, from the server end, it is now possible to force a client to disconnect and release its licenses. This can be useful in situations where a Pulsonix application has been left running on a remote computer, effectively 'blocking' those licenses from being used by someone else.

A new **administrator-level** command **Disconnect Client** has been added to the **Tools** menu in **NLServer** (server as application, or when run as server 'Monitor'). To use this, select the entry in the table on the **Connections** tab and choose the new menu command. Confirm the prompt, and the next time the client sends its 'still alive' signal to the server it will receive a response that instructs the client to disconnect.

The client will then behave as it would if the network connection had been lost. When the user resumes working with the client application, if it cannot re-establish connection to the server then the user will be able to save any open files before closing and restarting the application, after which the licenses will be released back to the pool. On the client end, the application will go into the 'network disconnected' mode; they will still be able to save their work, then close and restart the application to acquire a new set of licenses.

Technology Files

Changes to existing Technology files

All existing technology files (SCM and PCB) have been enhanced to check and remove previous style inconsistencies'.

New Technology files

A new technology file named **flexirigid4.ptf** has been added. This can be used when creating a flexi-rigid PCB of 4 layers; two 'normal' electrical layers and two electrical flexi layers.

Library Changes

New Library Parts Added

New Library Parts have been added to the MasterLibraries for TDK filters.

The existing Hamlin parts have been modified reflecting manufacturers Part numbers, technologies and footprint areas. Footprints have been amended or created for the new Hamlin Parts added. 65 new Hamlin Parts for their range of SMT reed switches have been added.

The existing Omron parts have been modified and updated; adding full Part Numbers and descriptions. New Parts have been added totalling 152 library Parts.

New Doc Symbols Added

New Doc Symbols have been added to both the Schematic Doc Symbol and PCB Doc Symbol libraries.

Schematic Doc Symbols for **Left**, **Right** and **Both Pointer** types that include the new dynamic net naming feature as a callout. New symbols are also included for **alternative Drawing Borders** in Portrait and Landscape modes.

PCB Doc Symbols are included for general documentation purposes such as **CE-Mark**, **ESD**, **Draft** and '**For Information Only'** symbols.

Revised Footprint Libraries

Below is a summary of changes made to the footprint libraries:

- Existing footprint libraries have been removed and replaced with more logically named alternatives. All existing footprints have been retained and library sets have been added to, *see below*.
- Rationalisation of areas on chip resistors, capacitors, diodes and inductors. Placement Areas have been modified to bring more into line with realistic placement areas required.
- Addition of more complete sets of chip resistors, capacitors, diodes and inductor series.